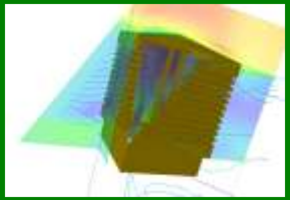
Computer Simulation of Fluid Flow, Heat Flow, Chemical Reactions and Stress in Solids.

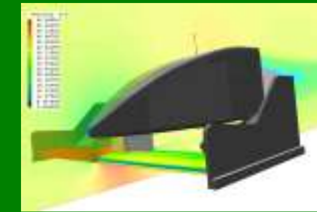


Seminar

CHAM

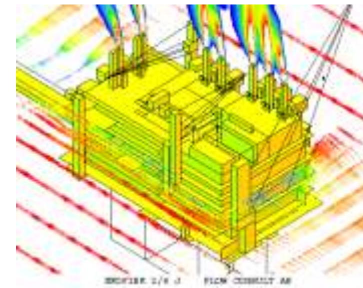
An introduction to CHAM, its software and services.

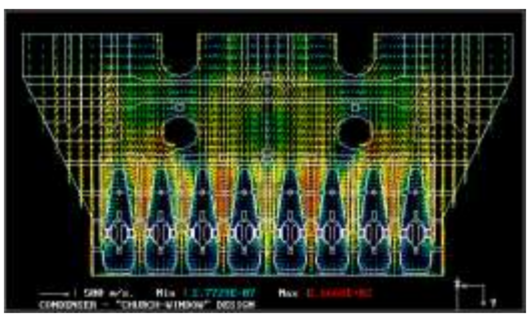
What does CHAM do?



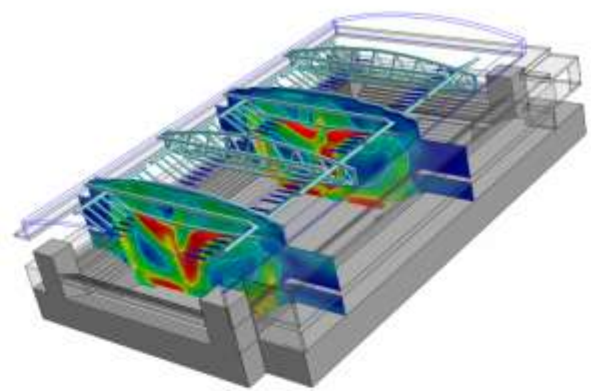
CHAM

"Simulation of processes involving fluid flow, heat transfer, chemical reaction & combustion within engineering equipment and the environment"



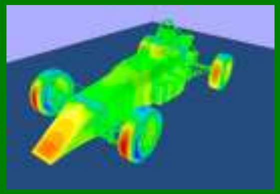


A-Z industrial & environmental applications



Practical - Cost-effective - Validated - Easy to Use

CHAM and its Services



Seminar

CHAM

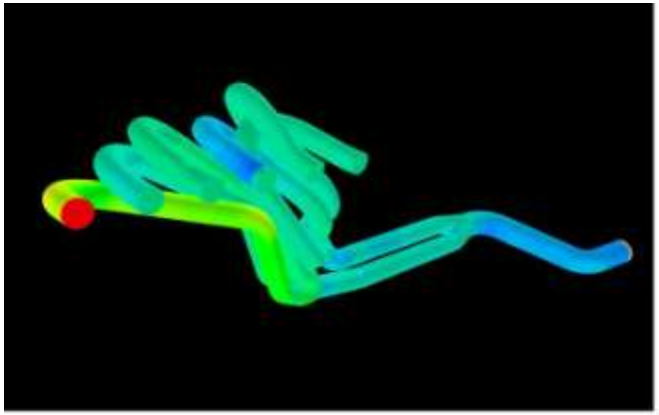
Research & Product Development

Special-Purpose Products

Customer and Technical Support

Consultancy Services

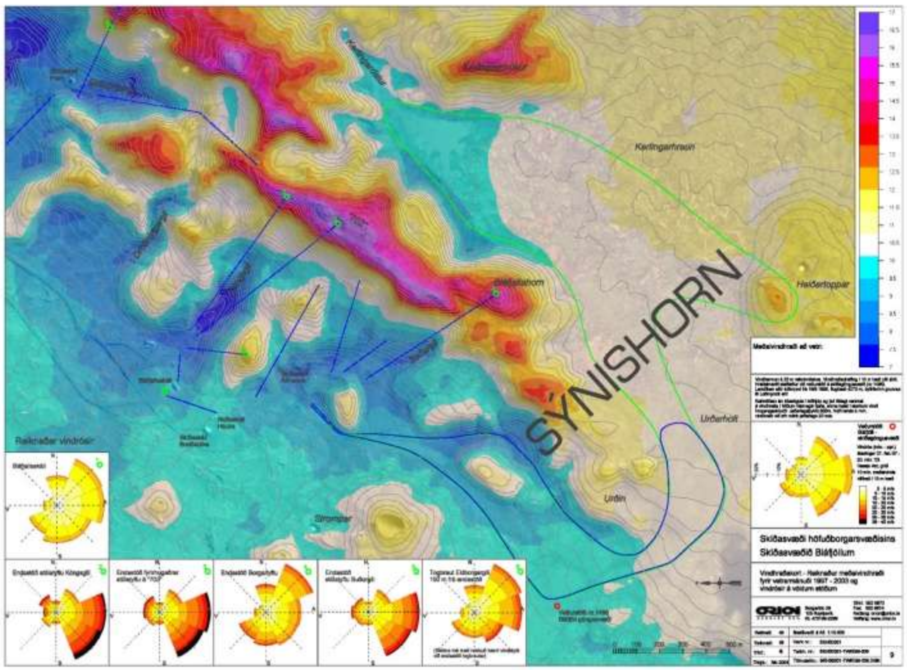
Model Build

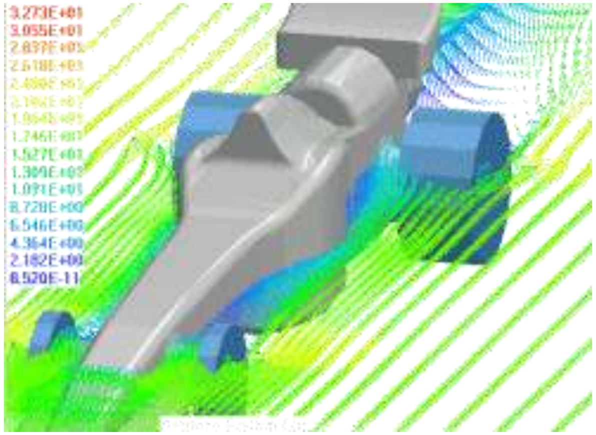


CHAM and its Partners

Seminar

CHAM



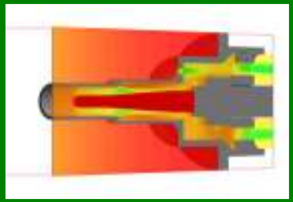


THE FORMULA ONE TECHNOLOGY CHALLENGE IN SCHOOLS

windsim

Bentley® Sustaining Infrastructure

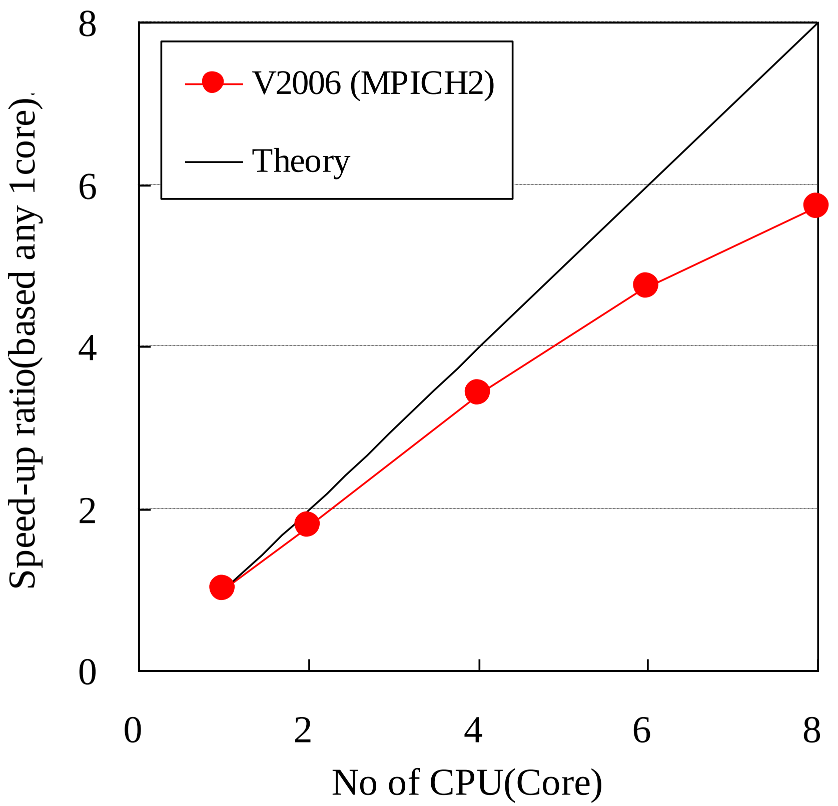
CHAM Technology Developments



CHAM

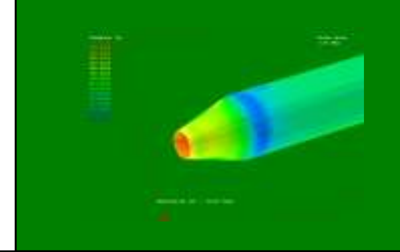
Parallel Processing achieves an average speed-up of 3.6 for cases run on quad-core systems.

CHAM supports Parallel PHOENICS on multi-core and HPC Windows & Linux clusters



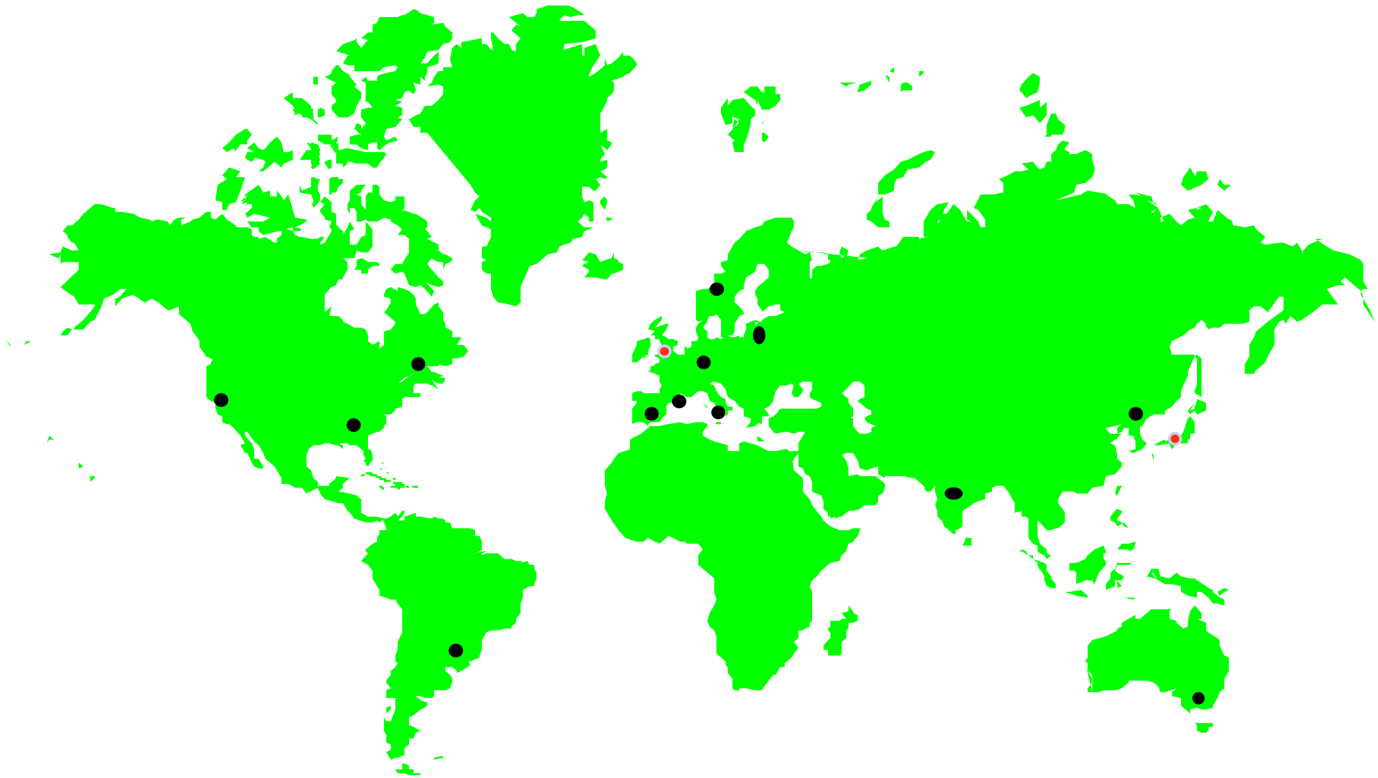
Case with 20 million cell mesh

CHAM Today



Seminar

CHAM

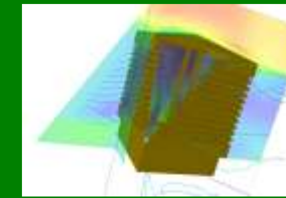


3 International offices

30 International representative agencies

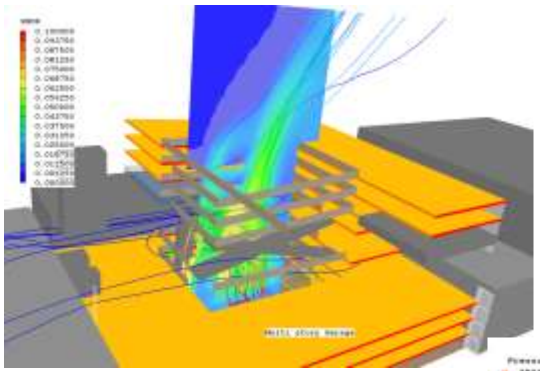
>4,000 PHOENICS customers world-wide

A wide range of applications



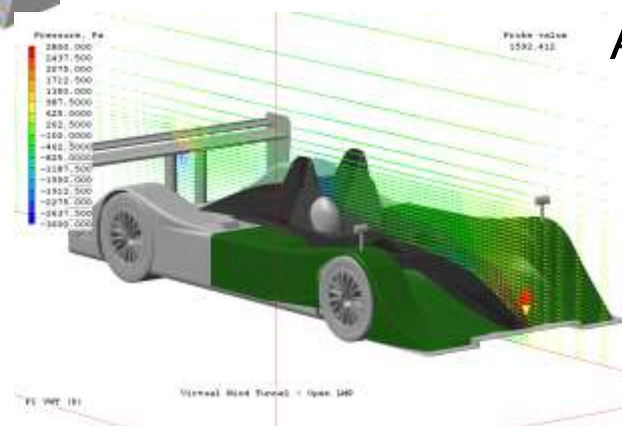
Seminar

CHAM

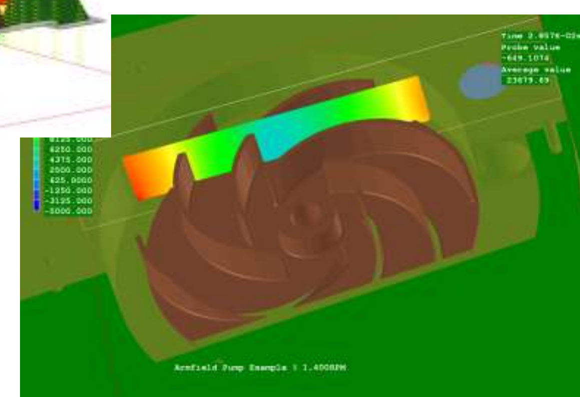


Smoke & fire spread

Heating & Ventilation



Aerodynamics

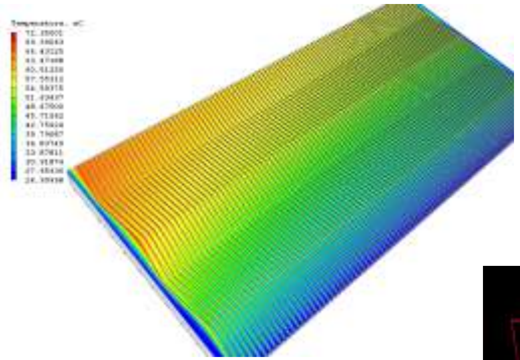


Impeller pump efficiency

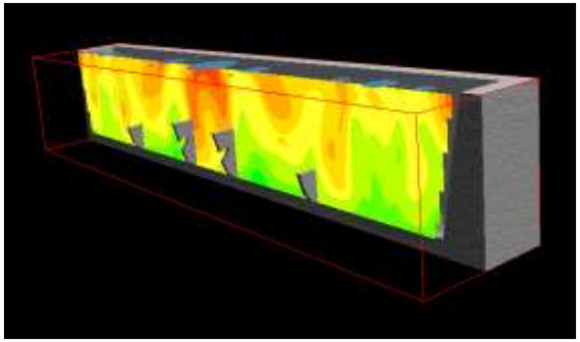
A wide range of applications

Seminar

CHAM

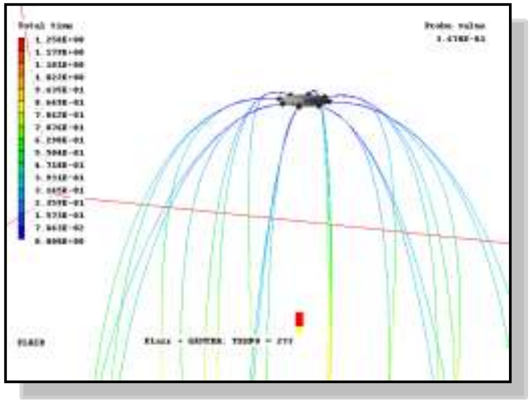


Heat transfer

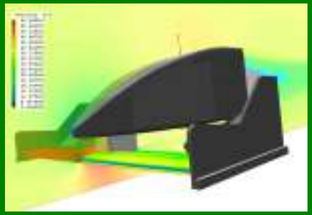


Multi-phase

Particle tracking



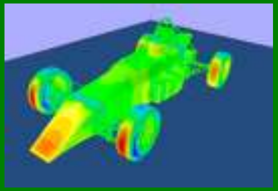
Main Features of PHOENICS



CHAM

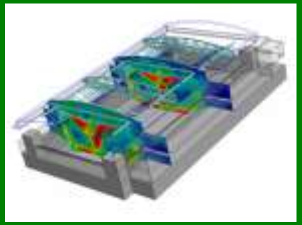
- 1-,2- and 3-D geometries
- Cartesian, Polar, Body-Fitted Coordinates, and Unstructured
- Local multi-level fine-grid embedding
- “PARSOL” Cut-cell technique for complex geometry
- “INFORM” Input of user-defined Formulae
- Conjugate Heat Transfer
- Single or Multi-Phase Flow
- Particle Tracking
- Chemical reaction
- Radiation
- Non-Newtonian Flow
- Choice of equation solvers and differencing schemes
- Automatic generation of user code
- Open-source routine for user-coding
- Automatic convergence control

Main Features of PHOENICS



CHAM

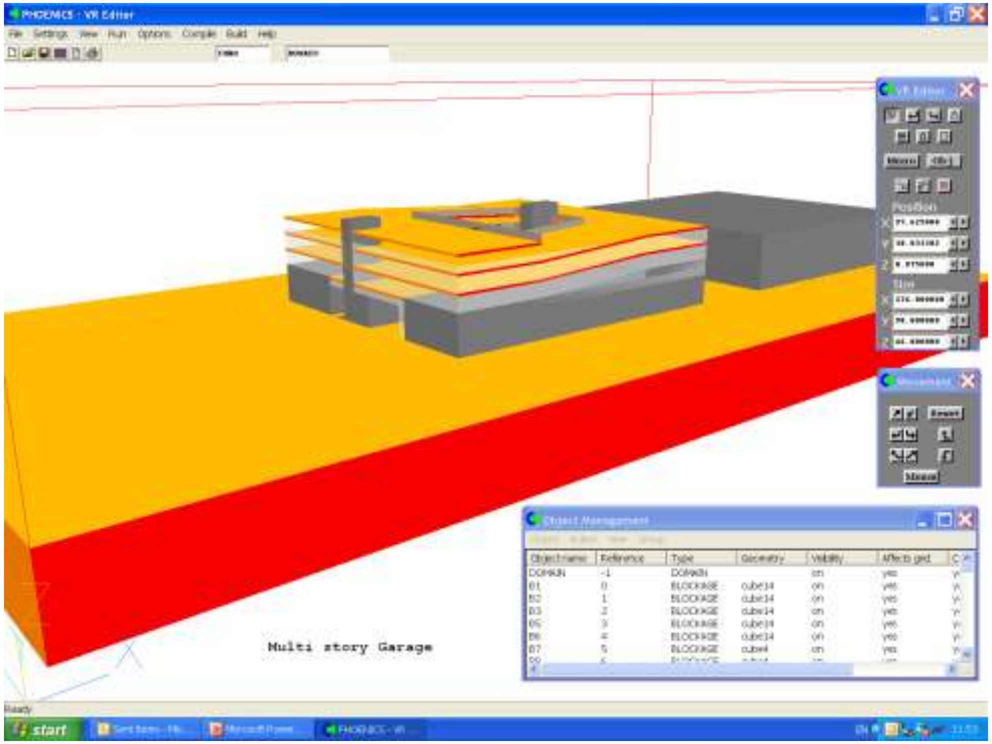
- PHOENICS consists of several modules:
  - **VR-Editor** for setting up problems,
  - **EARTH** for solving the problem,
  - **VR-Viewer** for visualising results; and
  - **POLIS** for providing information.
- Together they allow users to solve a wide range of 1D, 2D and 3D fluid flows simultaneously with heat transfer and chemical reaction.



CHAM

- Model setup – VR Editor

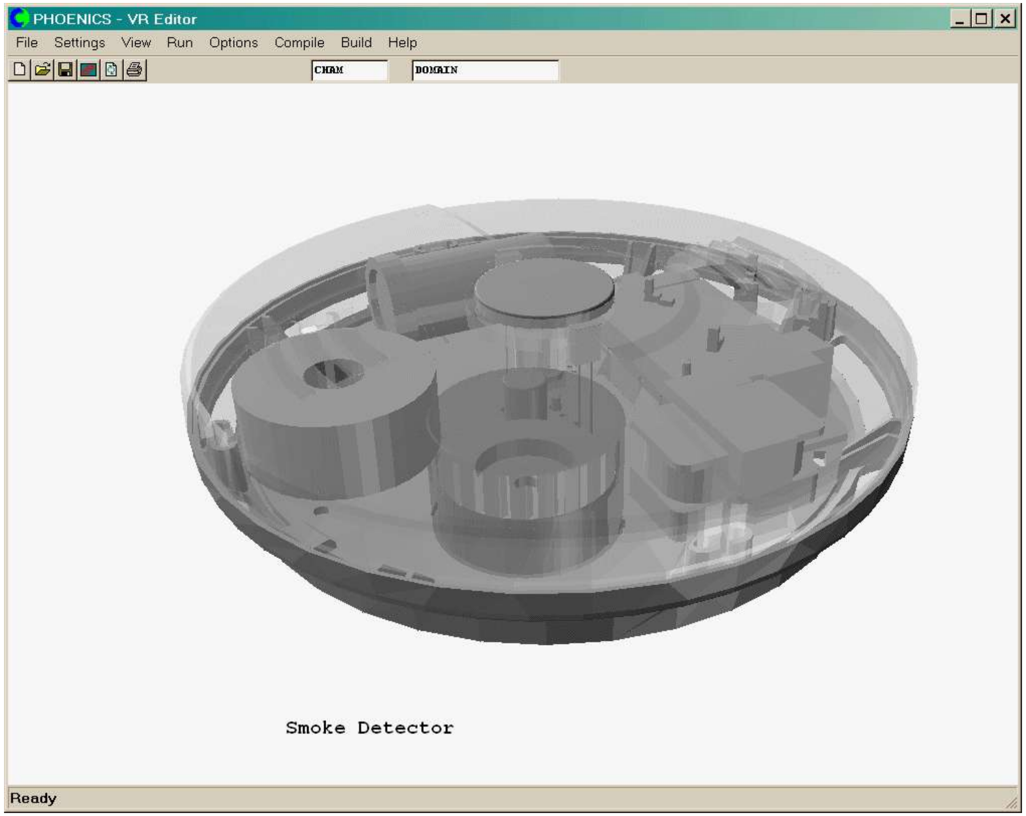
- Clicking on an object brings a dialogue box onto the screen.
- This enables the information about the object to be edited.

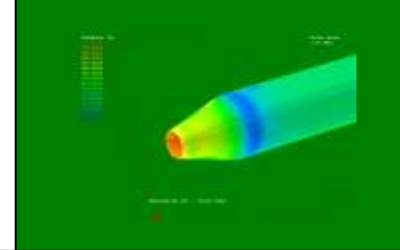


PHOENICS Key Components

CHAM

- Model setup – VR Editor
- The object geometry can be taken from a library of shapes, or loaded from a CAD file.
- CAD geometries are read using the STL format and many more besides





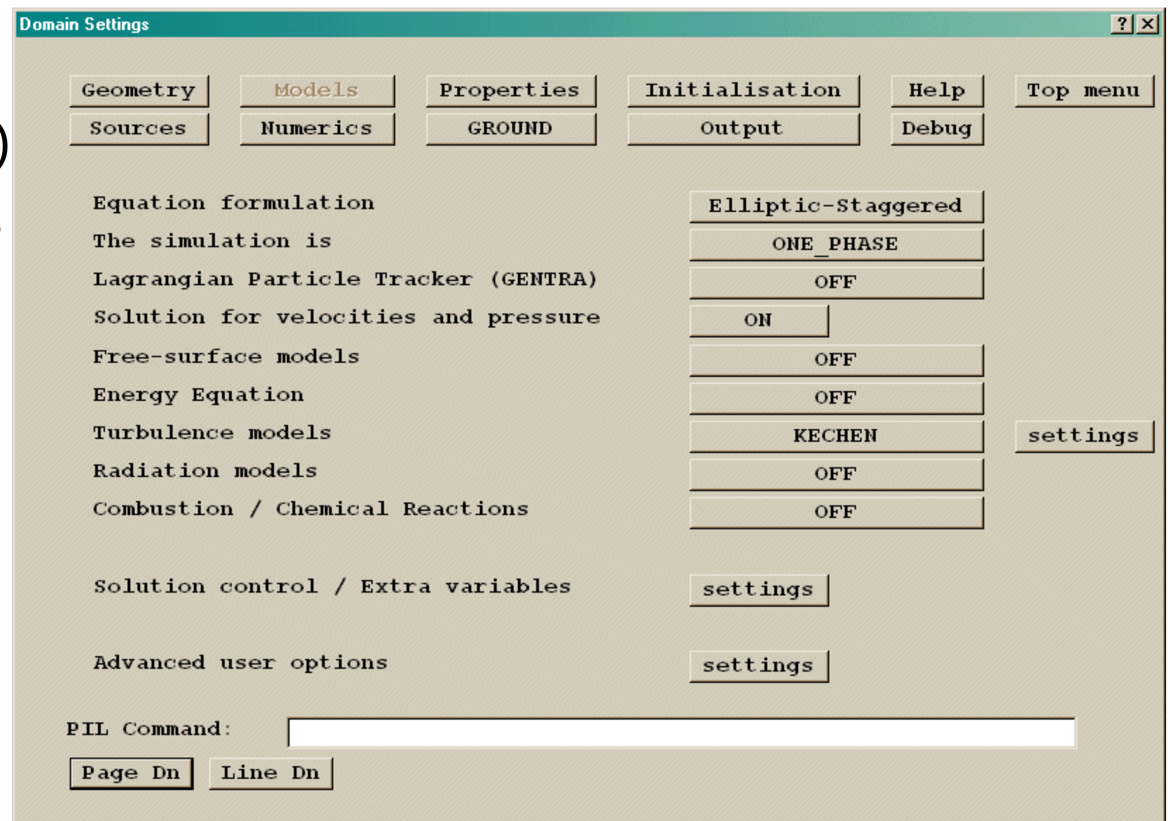
Setting Up Problems: PHOENICS-VR

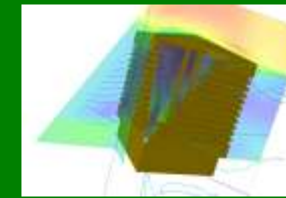
Seminar

CHAM

The PHOENICS-VR Main menu allows you to make all the settings required for a problem, including:

- Geometry
- Variables to be solved (models)
- Fluid properties
- Initial values
- Boundary conditions
- Monitoring options

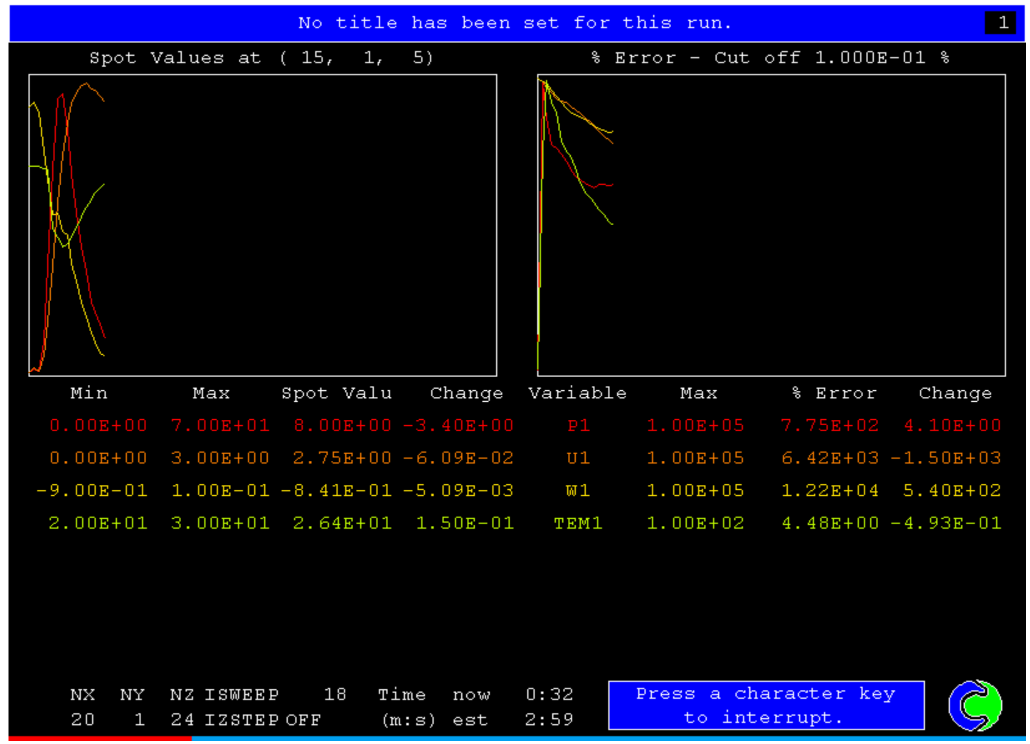


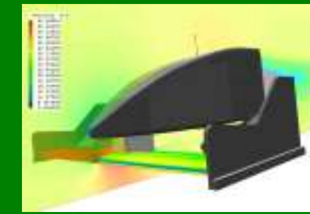


PHOENICS Key Components

CHAM

- Model setup - VR Editor
- Calculations – EARTH Solver
- EARTH is the program that performs the simulation.
- The graphical monitor shows the converging solution





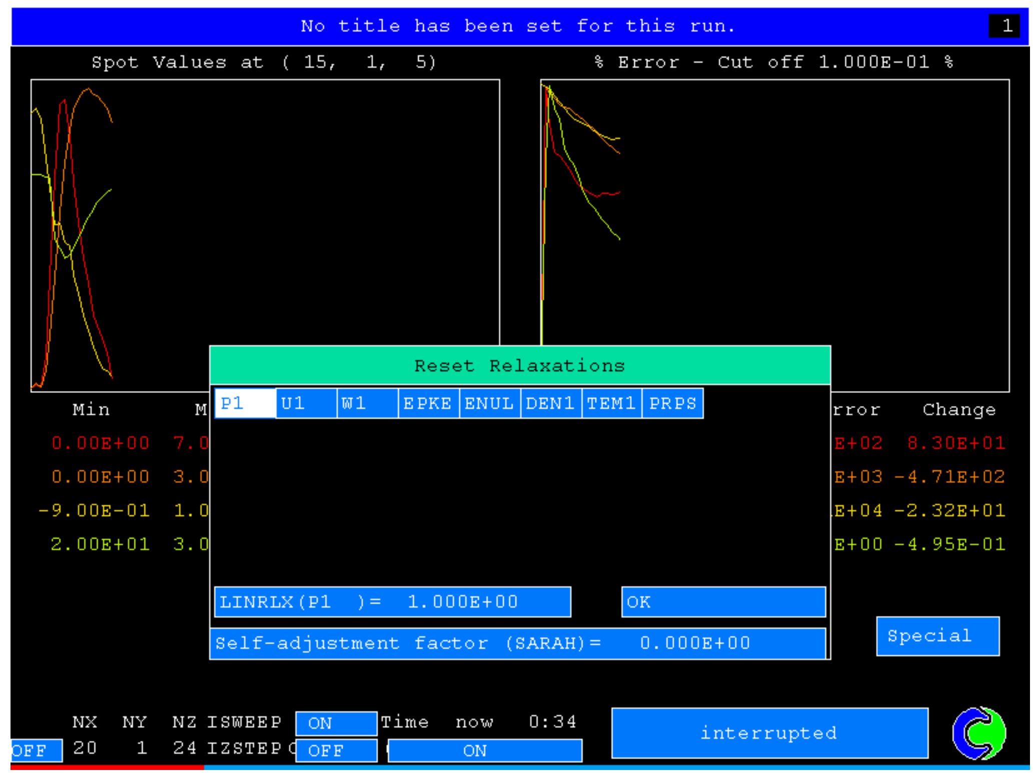
Calculations - Solver

CHAM

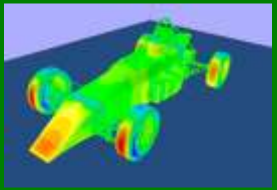
The EARTH run can be interrupted to change several parameters:

- Monitoring position
- Relaxation factors
- Graphical monitor settings

Intermediate result files can also be dumped

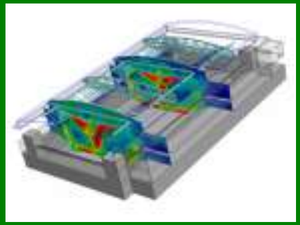


Calculations - Solver



CHAM

- EARTH is supplied partially as compiled object code, partially as open source code.
- The object code contains:
  - The basic solution algorithm and equation solvers
  - The input/output sequences
- The open source code contains:
  - The built-in turbulence, combustion, radiation and other physical models
  - The built-in physical property variations
  - The built-in higher-order differencing schemes



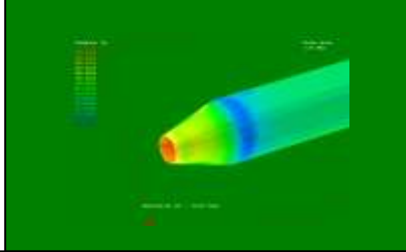
Calculations - Solver

- Part of the EARTH open source is the user-routine **GROUND**.
- This enables users to add in any new models, properties, source terms, or input/output sequences they may wish.
- Users not wishing to write code themselves may place algebraic expressions into the input file via “**INFORM”**
- These will be automatically executed at run-time. In this case, no compilation or linking is required!

PHOENICS Key Components

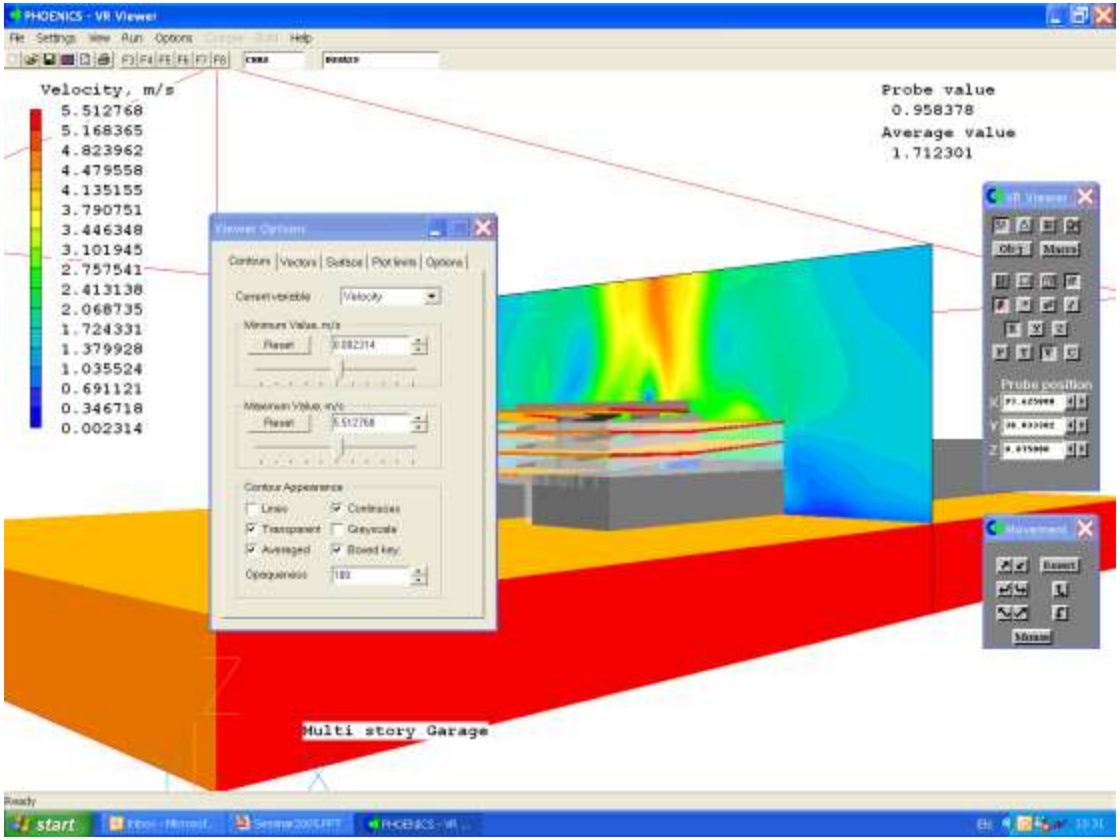
- Model setup – VR Editor
- Calculations - Solver
- Analysis of results – VR Viewer

Analysis of Results - VR Viewer

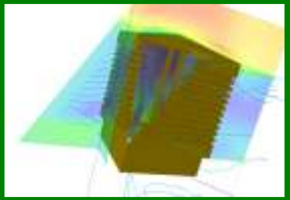


CHAM

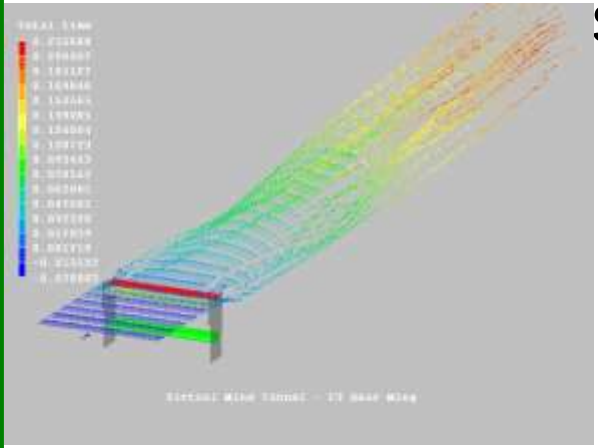
The VR Viewer allows users to see their results in a number of different ways:



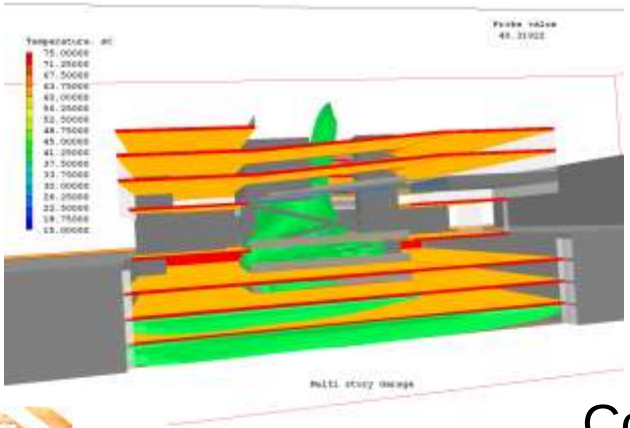
Analysis of Results - VR Viewer



CHAM

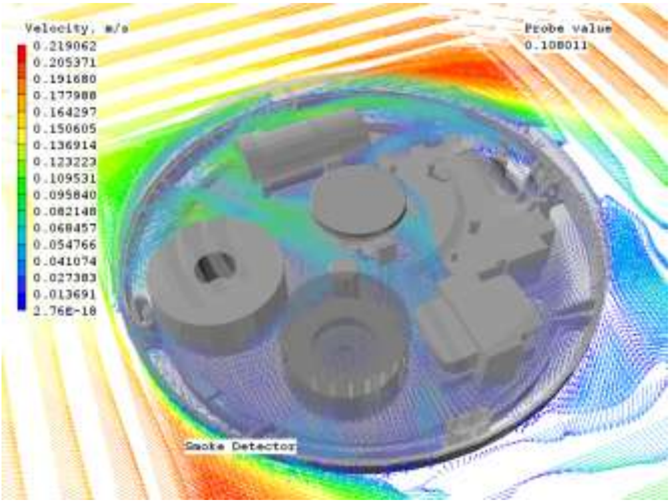


Streamlines, static or animated

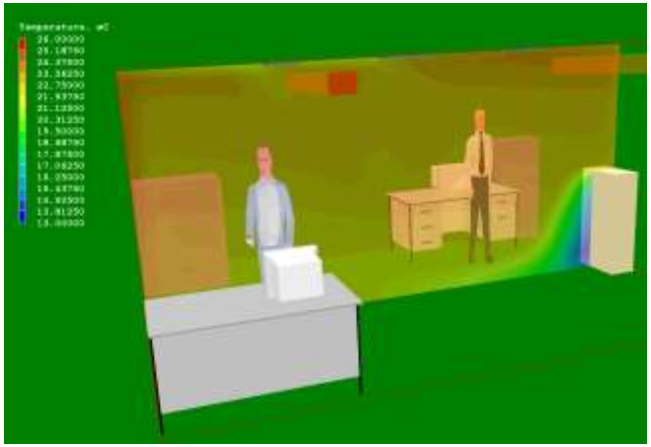


Iso-surfaces

Contour planes



Vectors



How to obtain PHOENICS

Licensing Options

- Monthly, Annual or Perpetual
- 32-bit & 64-bit sequential- or parallel-processing
- Windows or Linux

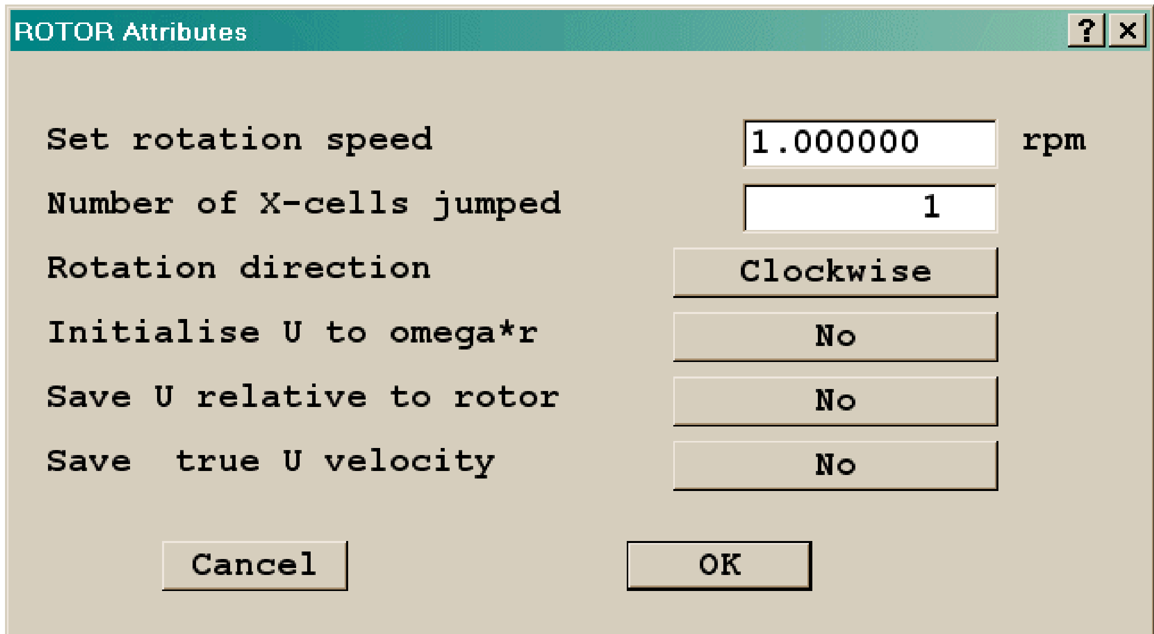
Special-Purpose Options

- FLAIR – HVAC, fire safety, building services
- VWT – Virtual Wind Tunnel
- ESTER – Electrolytic smelter
- CVD – chemical vapour deposition

Services

- Standard 3 day training course
- Day rate for consultancy and model build
- Extended consultancy support

Object type ROTOR introduces zone of rotating co-ordinate within static domain.



All cells within the rotor object are shifted in X relative to the static domain at the start of each time step.

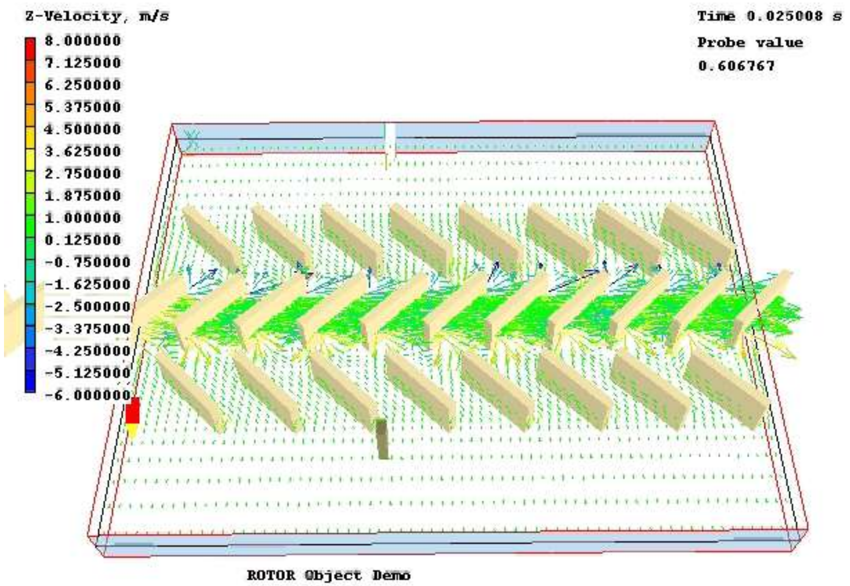
Only one object-detection sequence is needed, as everything within the rotor goes round with it.

PHOENICS Features
Rotating Co-ordinates

Seminar

CHAM

Time step size and number of steps are automatically set from the input parameters.

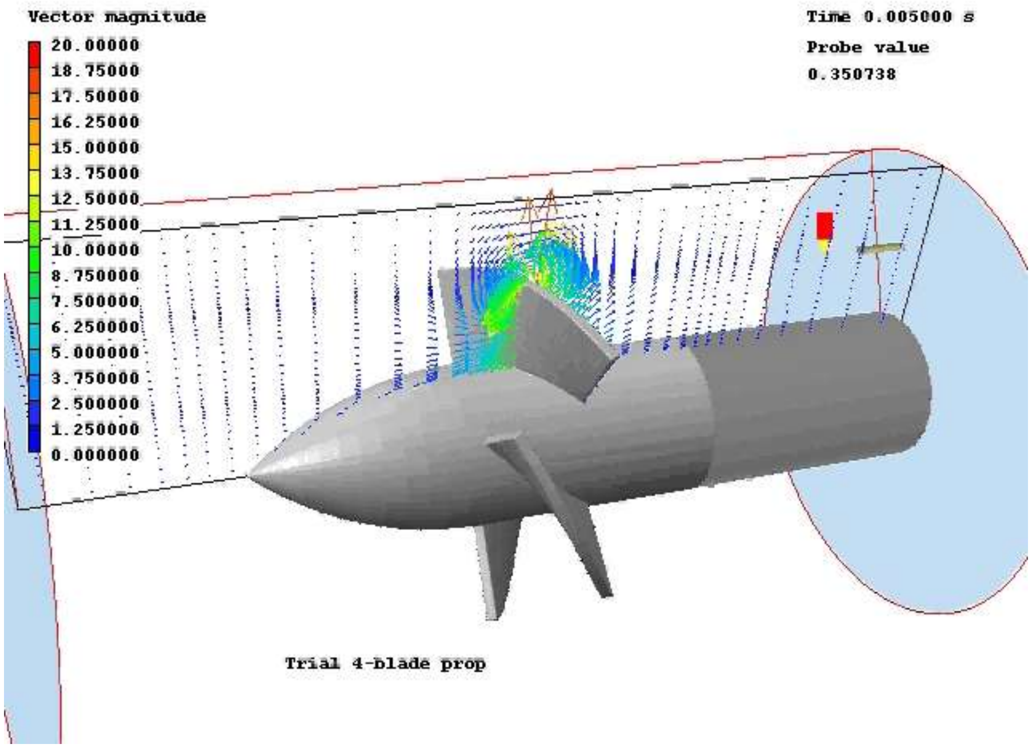


First example shows a row of simplified blades pulled between two simplified stators.

PHOENICS Features
Rotating Co-ordinates

Seminar

CHAM



Second example shows flow induced by a rotating propeller

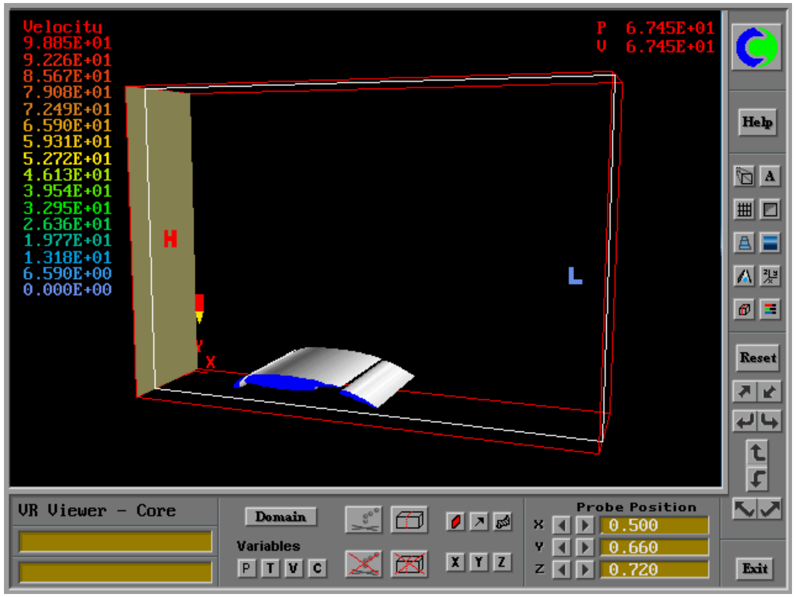
PHOENICS Features
PARSOL
(Cartesian cut-cell examples)

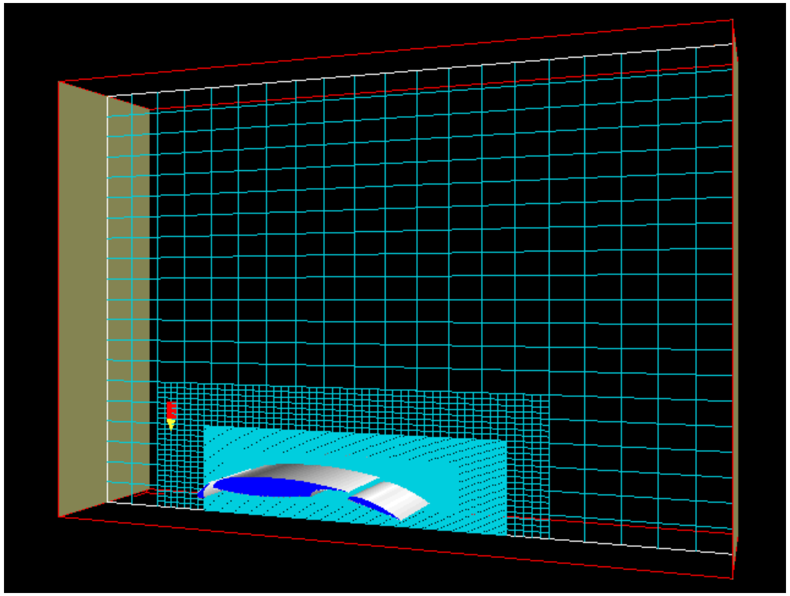
Seminar

CHAM

PARSOL and fine-grid embedding applied to a three-part airfoil

The flow is two-dimensional, incompressible, inviscid and steady.

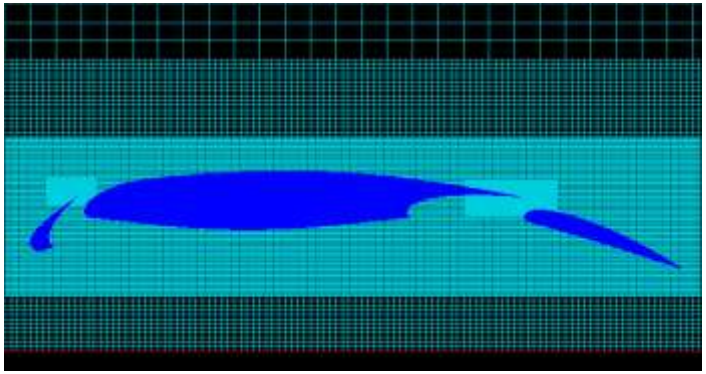




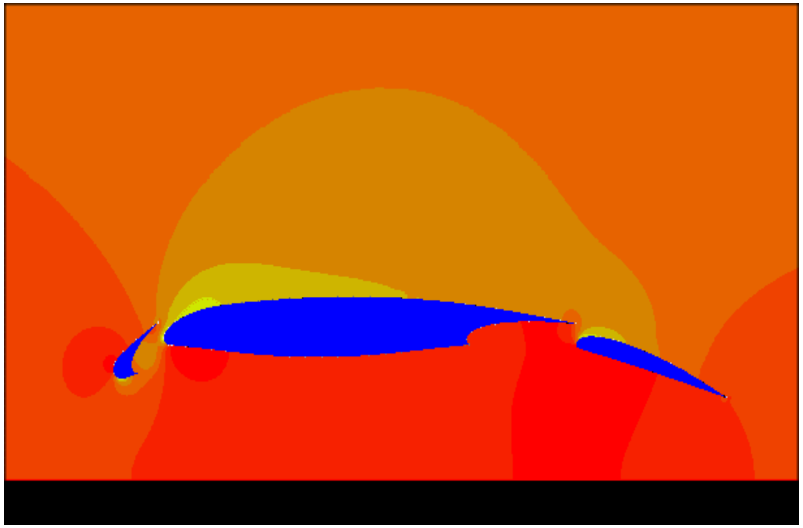
PHOENICS Features
PARSOL
(Cartesian cut-cell examples)

Seminar

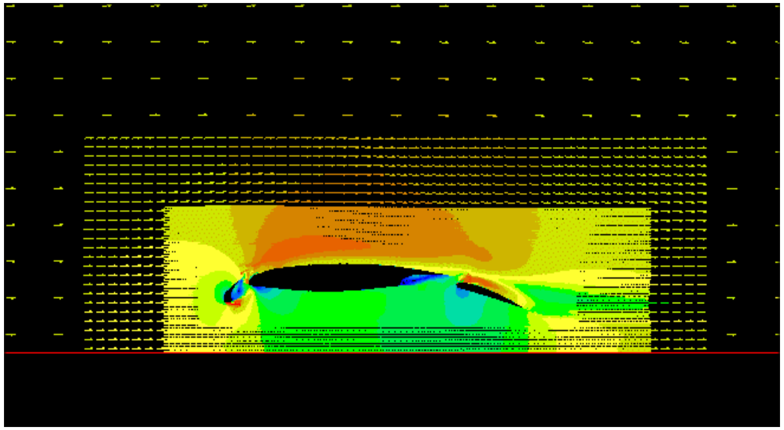
CHAM

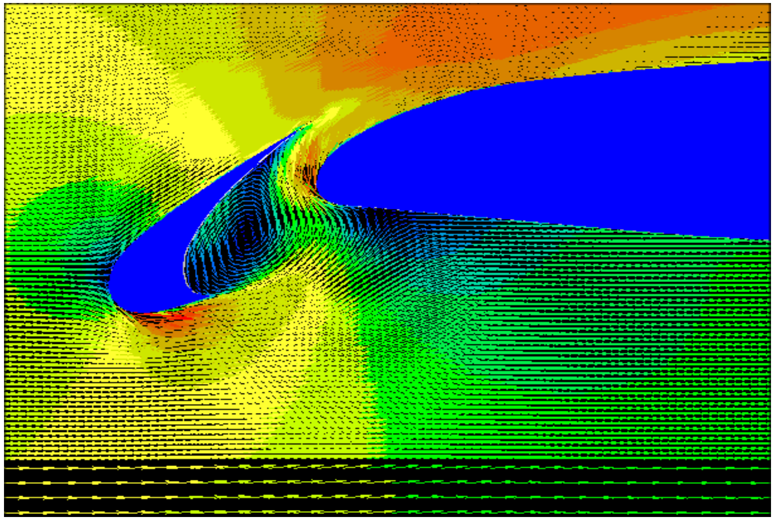


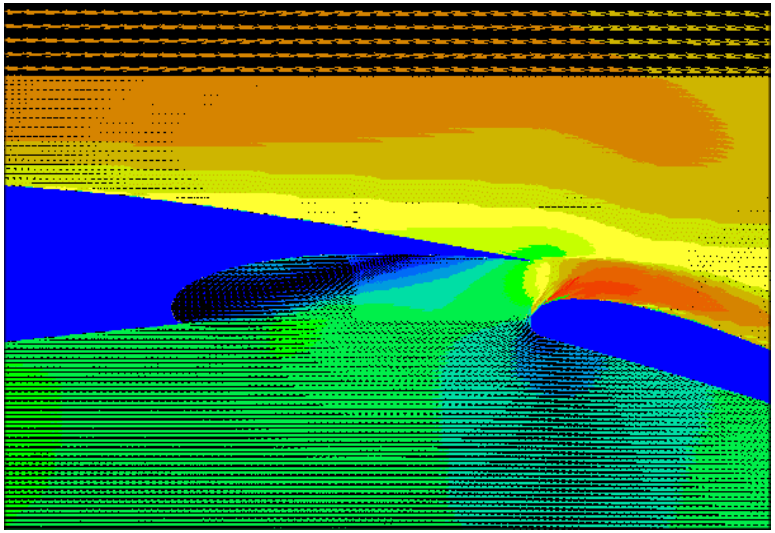
Embedded fine grids are used to capture airfoil detail



PHOENICS Features
PARSOL
(Cartesian cut-cell examples)







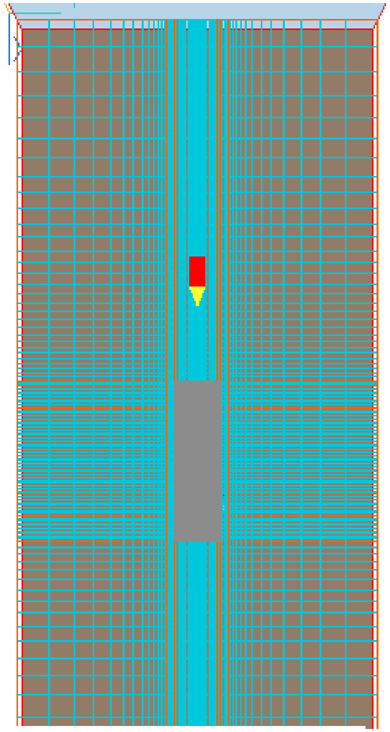
PHOENICS Features
Unstructured

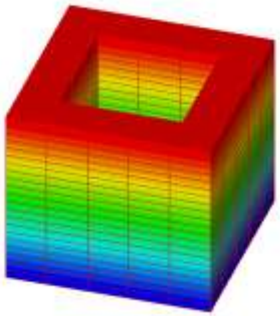
Seminar

CHAM

The **motive** for introducing PHOENICS Unstructured (**USP**) has **not** been (as it may be for competitors) to handle **curved-surface** bodies; for **PARSOL** handles these very satisfactorily.

Instead, the motive is to reduce the **time and storage** entailed by the un-needed fine-grid regions which PHOENICS (in structured- grid mode) generates far from the bodies, as seen on the right.



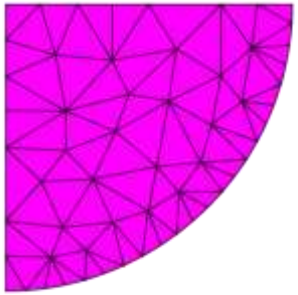


For the hollow-box heat-conduction problem on the left, **SP (structured PHOENICS)** pays attention also to the **empty central** volume; **USP does not.**

PHOENICS Features
Unstructured

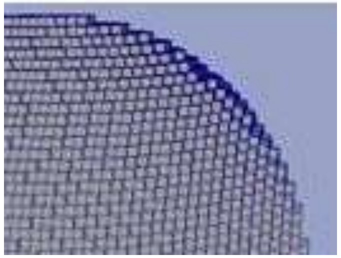
USP is a part of the standard PHOENICS package, which can therefore work in structured or unstructured modes at user's choice.

All USP grids consist of Cartesian (*i.e.)* brick-shaped cells. The general polygonal shapes such as this → used in other codes have been judged to be needlessly complex.

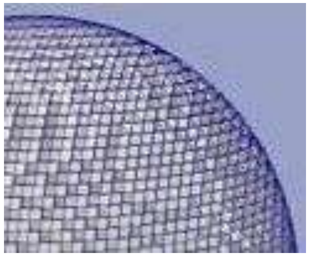


USP cells adjoining objects with curved surfaces can be distorted so as to fit them better, as shown on the right →

rectangular



distorted



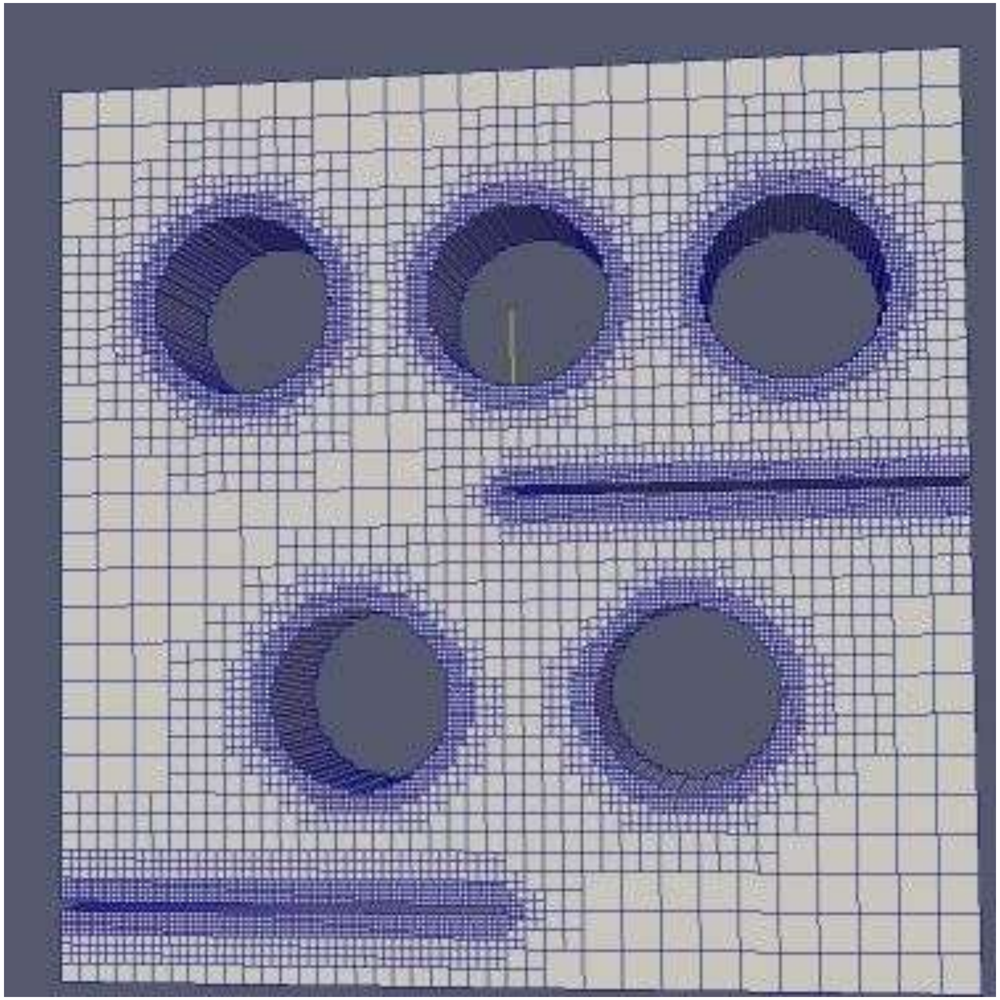
PHOENICS Features
Unstructured

Seminar

CHAM

This grid was created by means of **AGG**, the Automatic Grid Generator, a utility program which is supplied with the PHOENICS package.

AGG detects the presence, size and location of facetted 'virtual-reality' objects, and then fits layers of small cells to their surfaces.



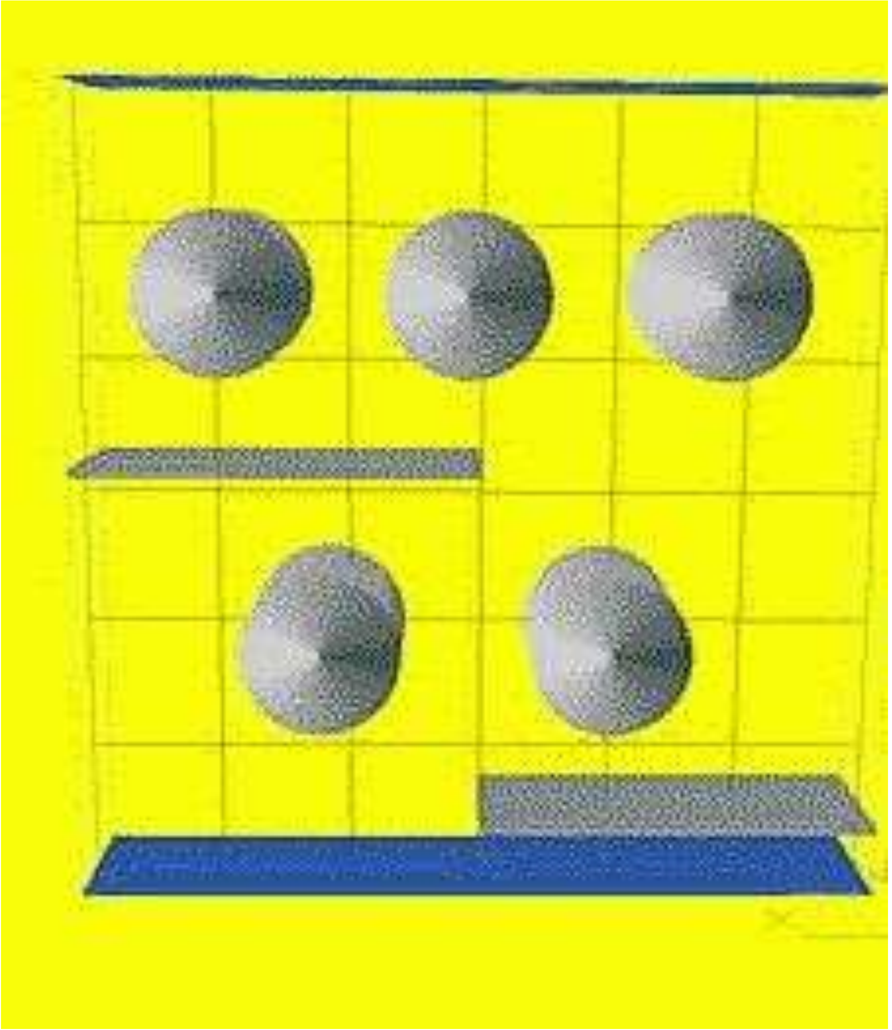
USP and AGG: Example #1
2D Heat conduction in plate with holes

A plate is perforated by holes and slots.

Heat is conducted from the top boundary at 10 degrees

to the bottom boundary at 0 degrees.

The **coarse grid** from which AGG starts is shown by the dark lines.



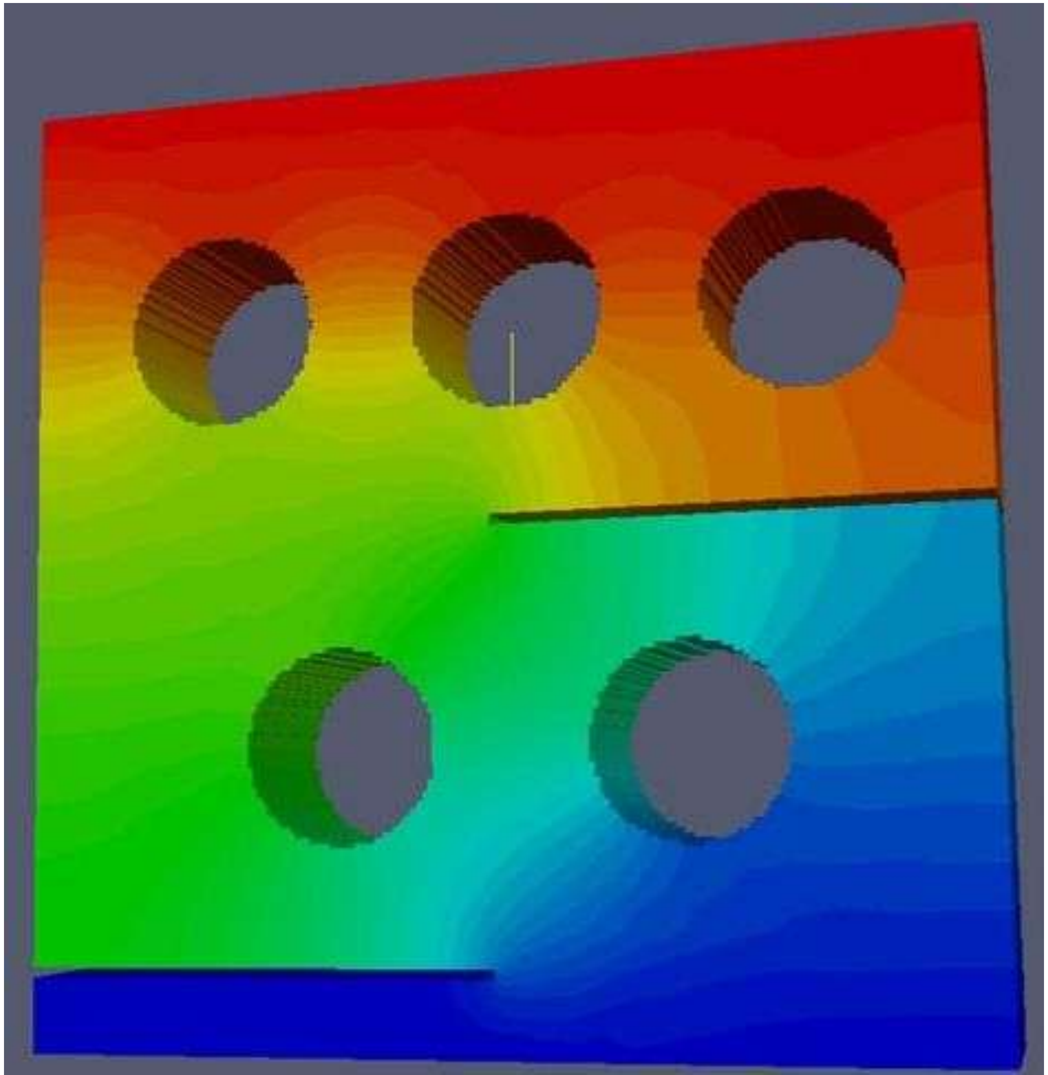
USP and AGG: Example #1
2D Heat conduction in plate with holes

CHAM

The resulting temperature contours reveal the expected effects:

the slots and holes serve as barriers to the flow of heat.

Of course, structured PHOENICS could have solved this problem easily with a uniformly fine grid, but at greater expense.



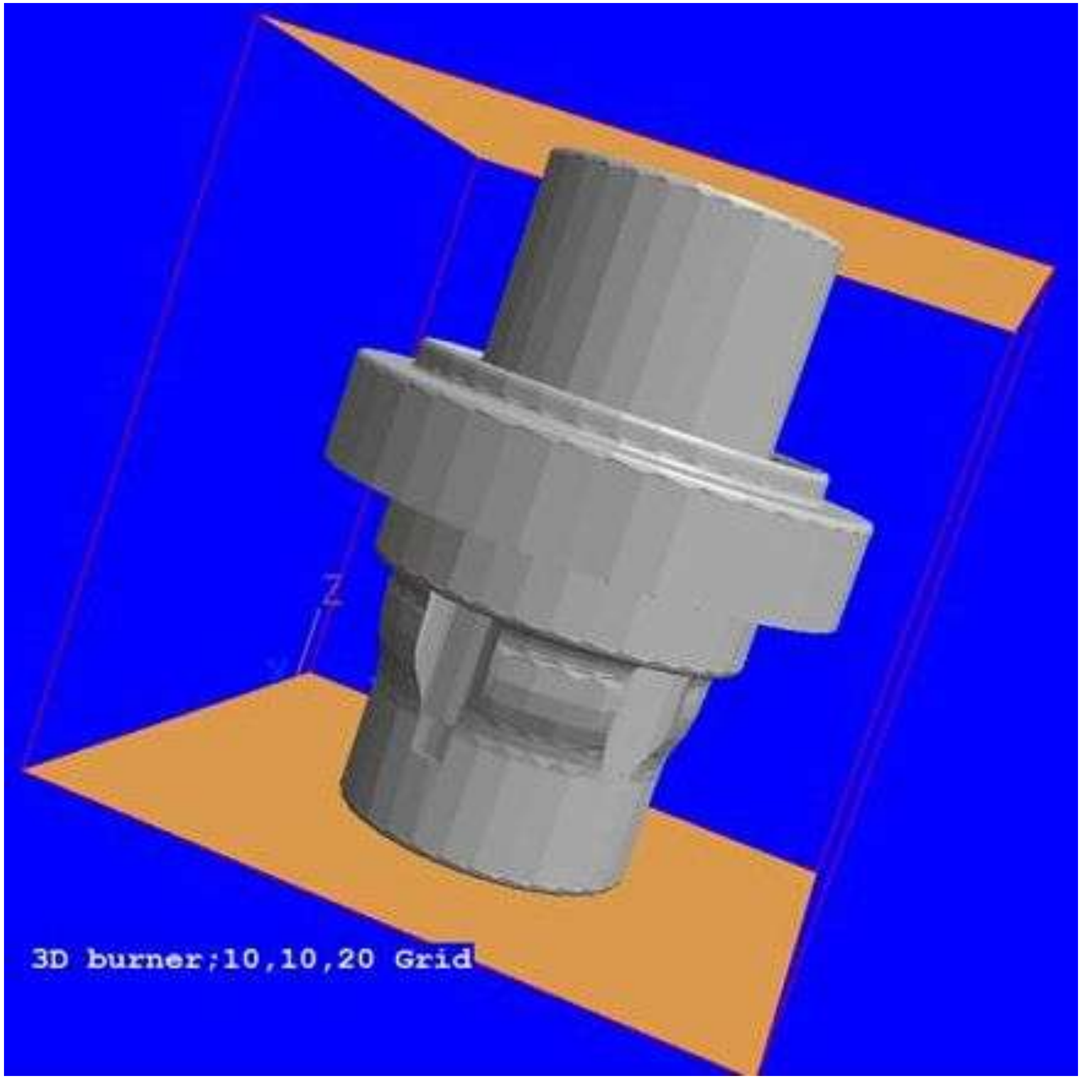
USP and AGG: Example #2; 3D Heat conduction

Seminar

CHAM

Heat flows from the bottom boundary of a hollow 3D object at 10 degrees C to the top boundary at 0 degrees C.

If SP were used:
- a fine grid would have to be used for the whole of the bounding-box space
- much of the computing time would have been wasted.



USP and AGG: Example #2; 3D Heat conduction

CHAM

On the right are shown the cells which touch the inner and outer surfaces of the solid body.

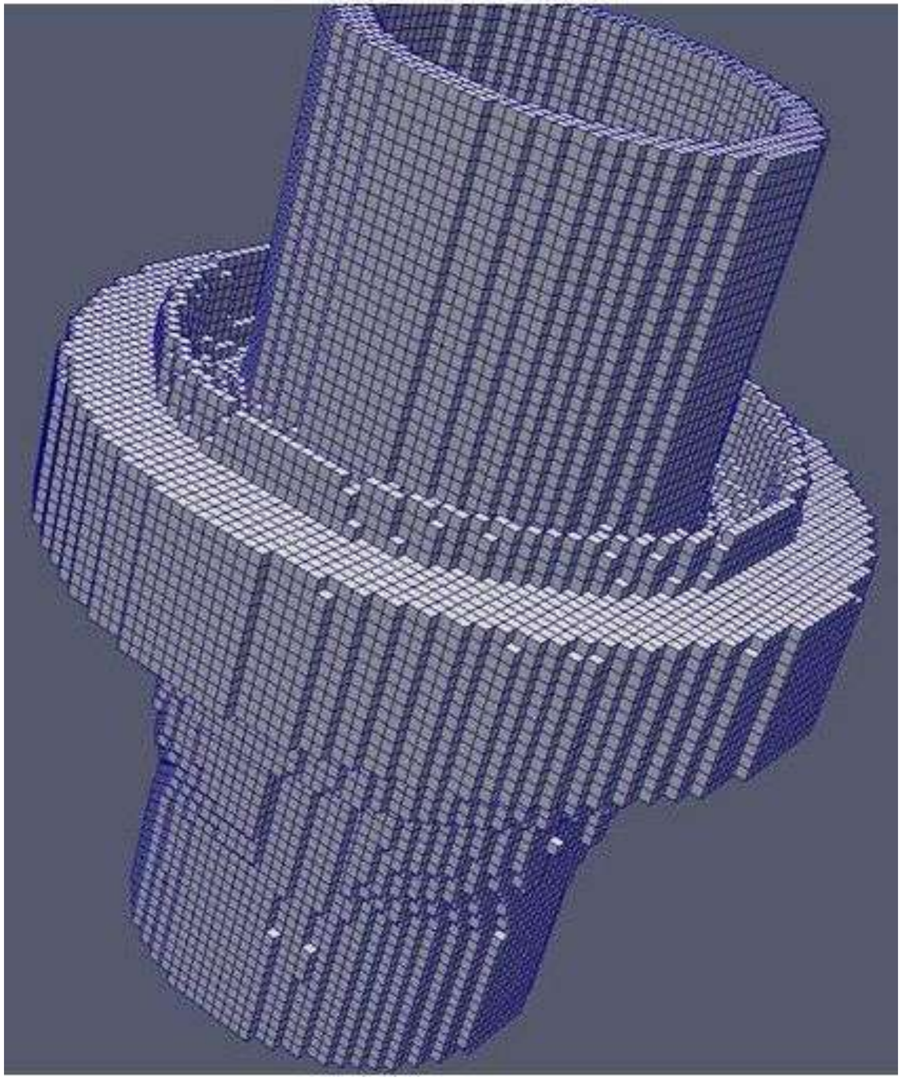
They are of a uniformly small size.

Larger cells fill the remainder of the volume of the object.

No cells exist at all in the non-solid spaces.

AGG has therefore built a grid of maximum economy.

Cell distortion for better fitting is **not** used here.



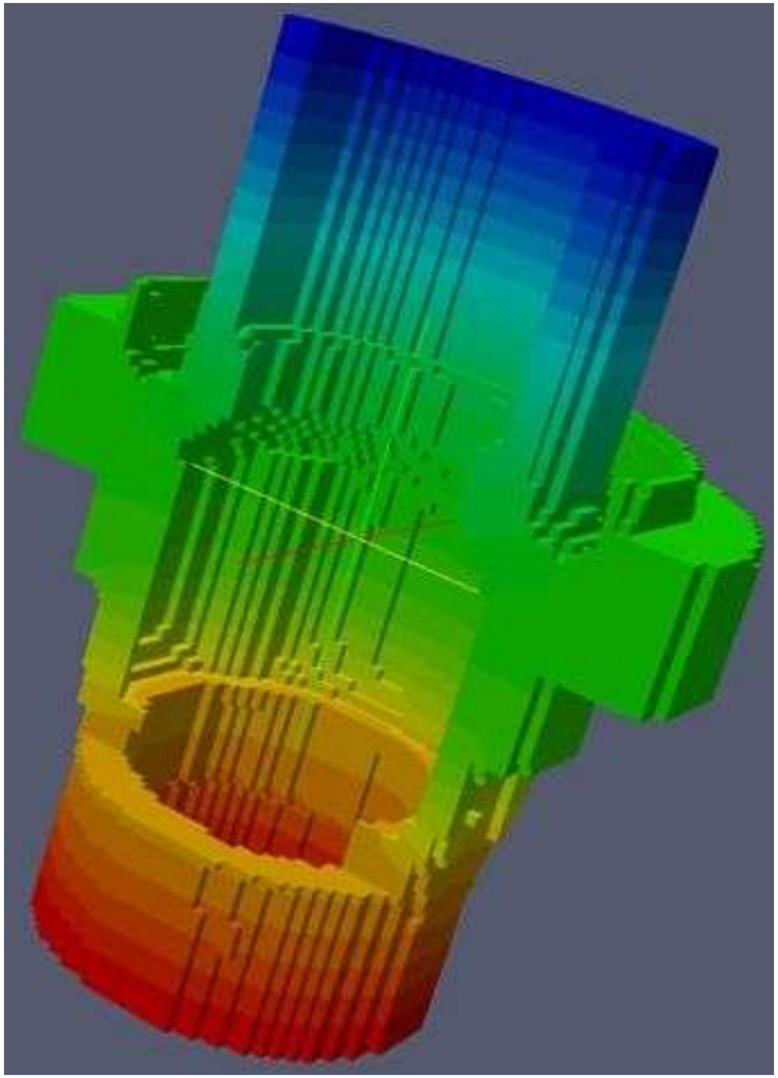
USP and AGG: Example #2
Computed temperature distribution

The **temperature contours** are shown on the right.

Part of the body has been cut away in order that the contours on the inner surface can be seen.

If there had been **fluid** inside and outside the body, AGG would have created cells in those regions also.

Then USP would have calculated the temperatures there too; and also velocities and pressures, **there only**.



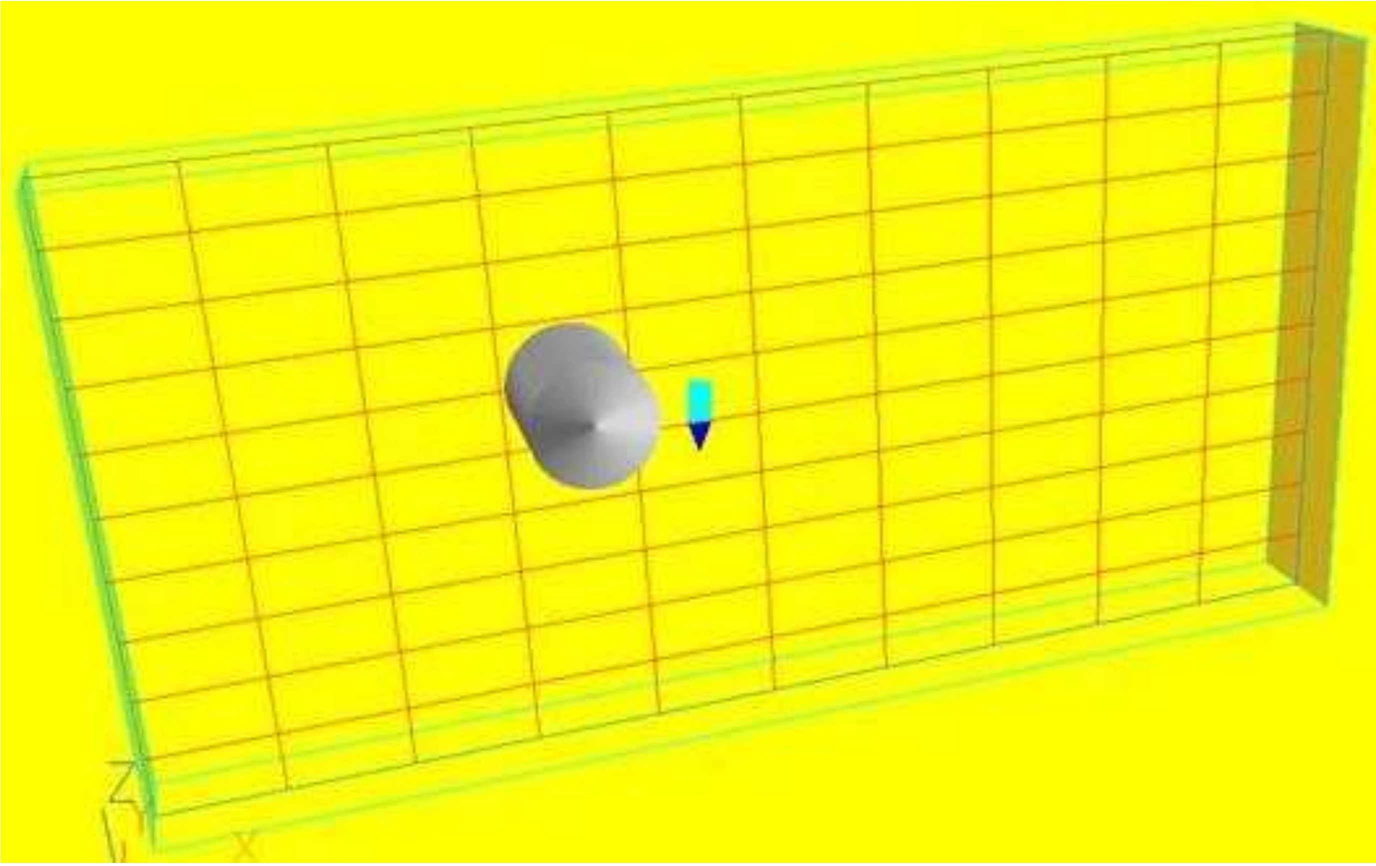
USP and AGG: Example #3; Flow around a cylinder

Flow is present in this third example which concerns steady laminar flow around a cylinder within a duct of finite width, from left to right.

The geometry is 2D.

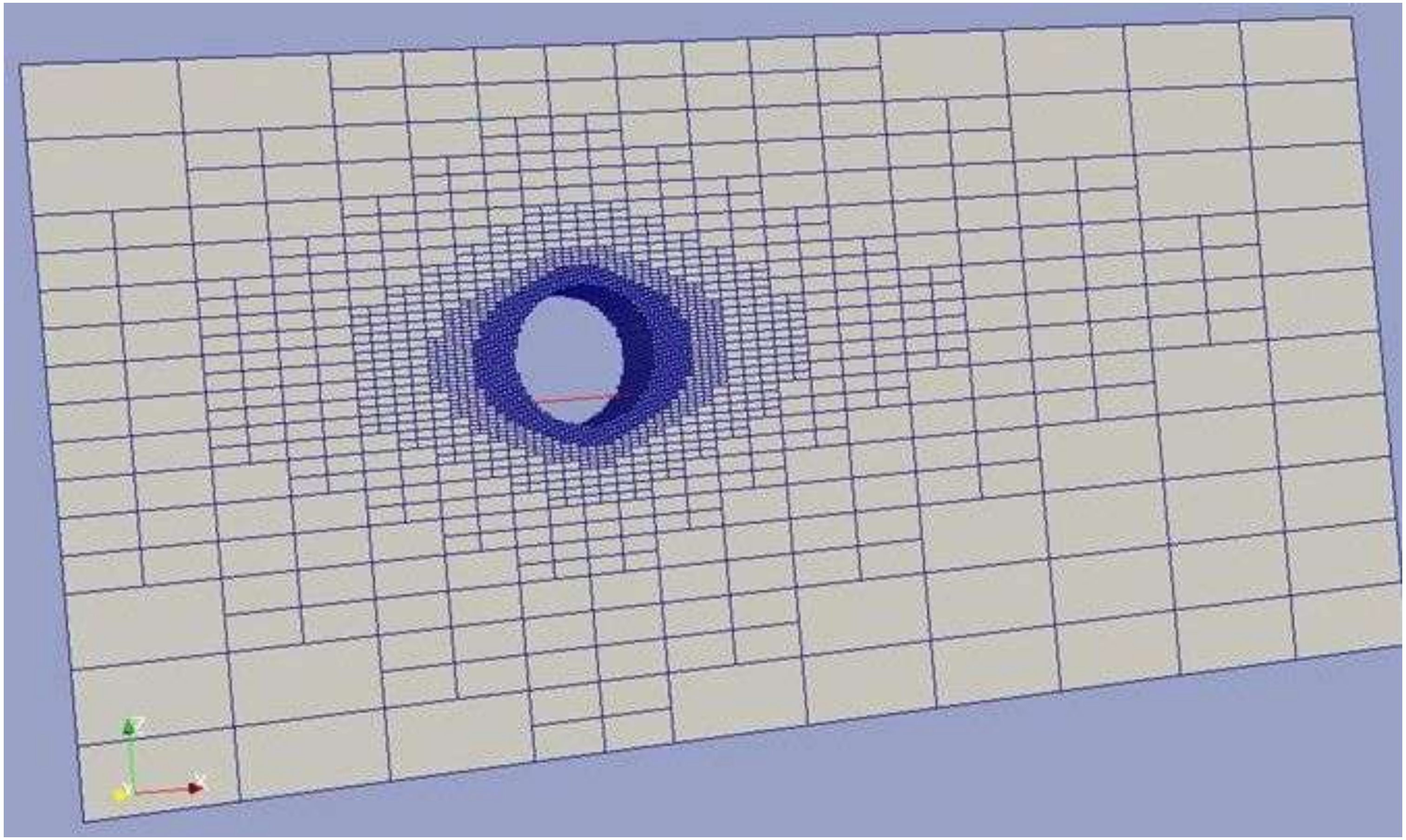
The Reynolds number is 40.

AGG starts with the coarse grid.



USP and AGG
Example #3; the unstructured grid

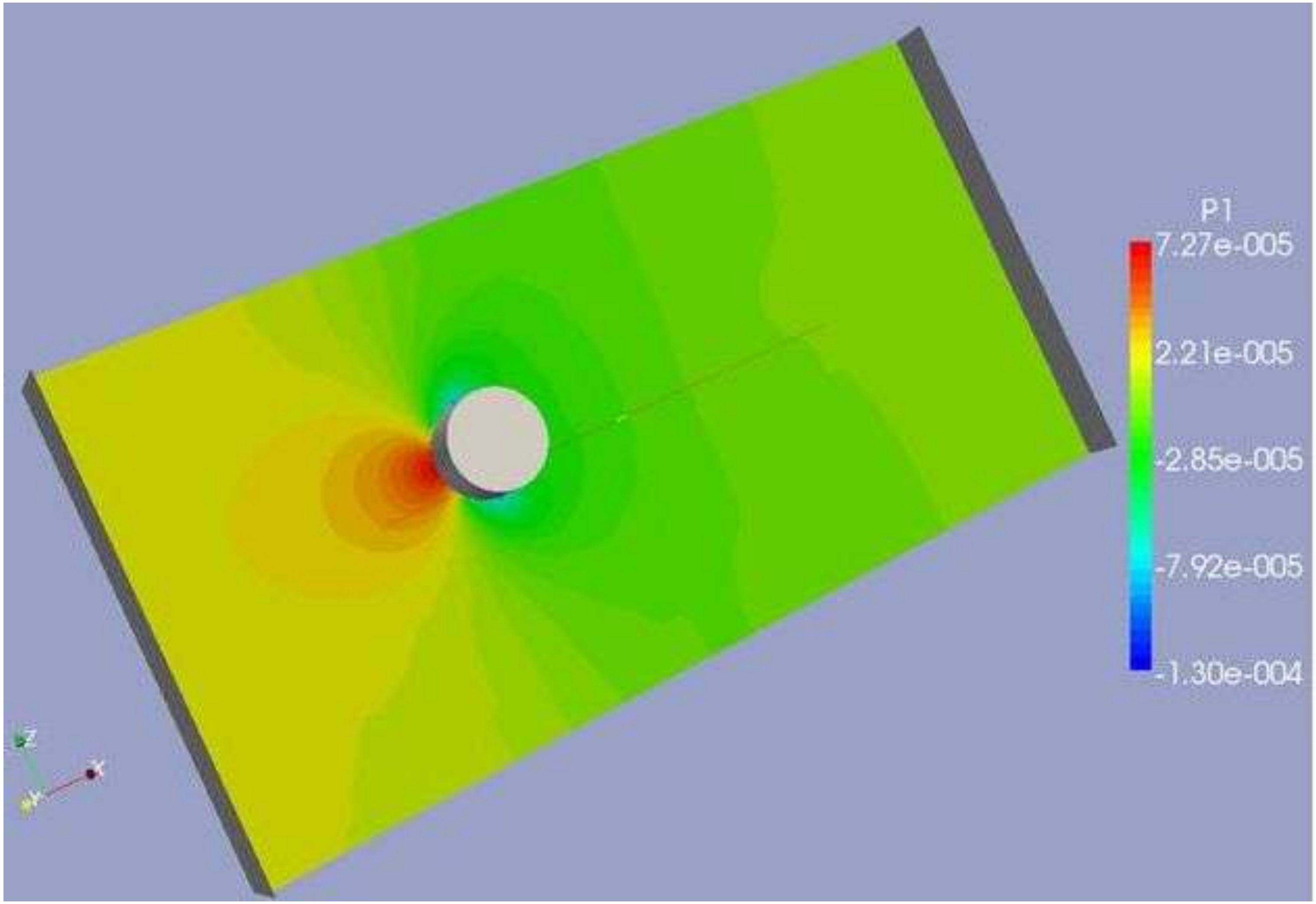
AGG created this grid, with **smallest** cells nearest to the **surface**



USP and AGG: Example #3
Computed pressure contours

Seminar

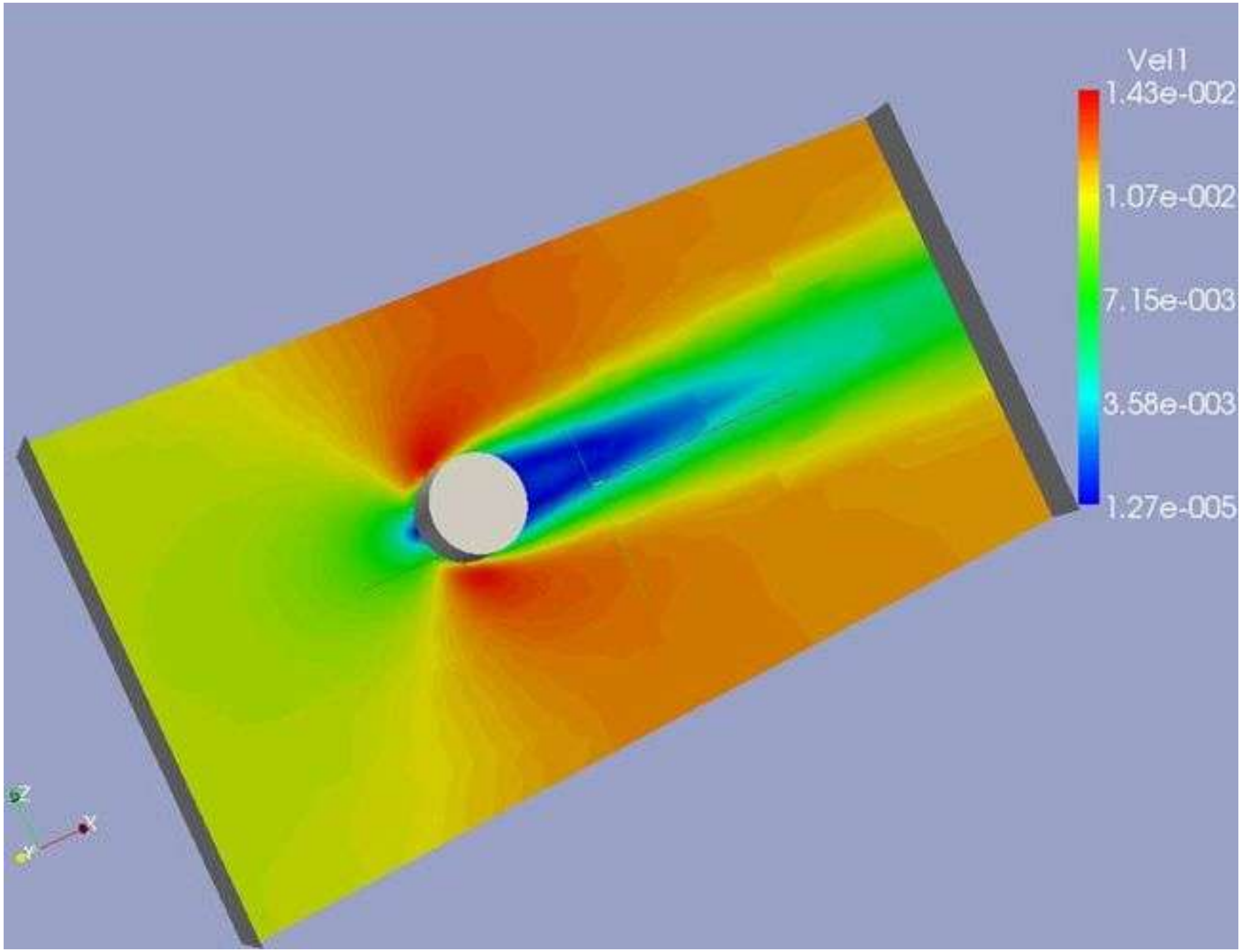
CHAM



USP and AGG: Example #3
Computed velocity contours

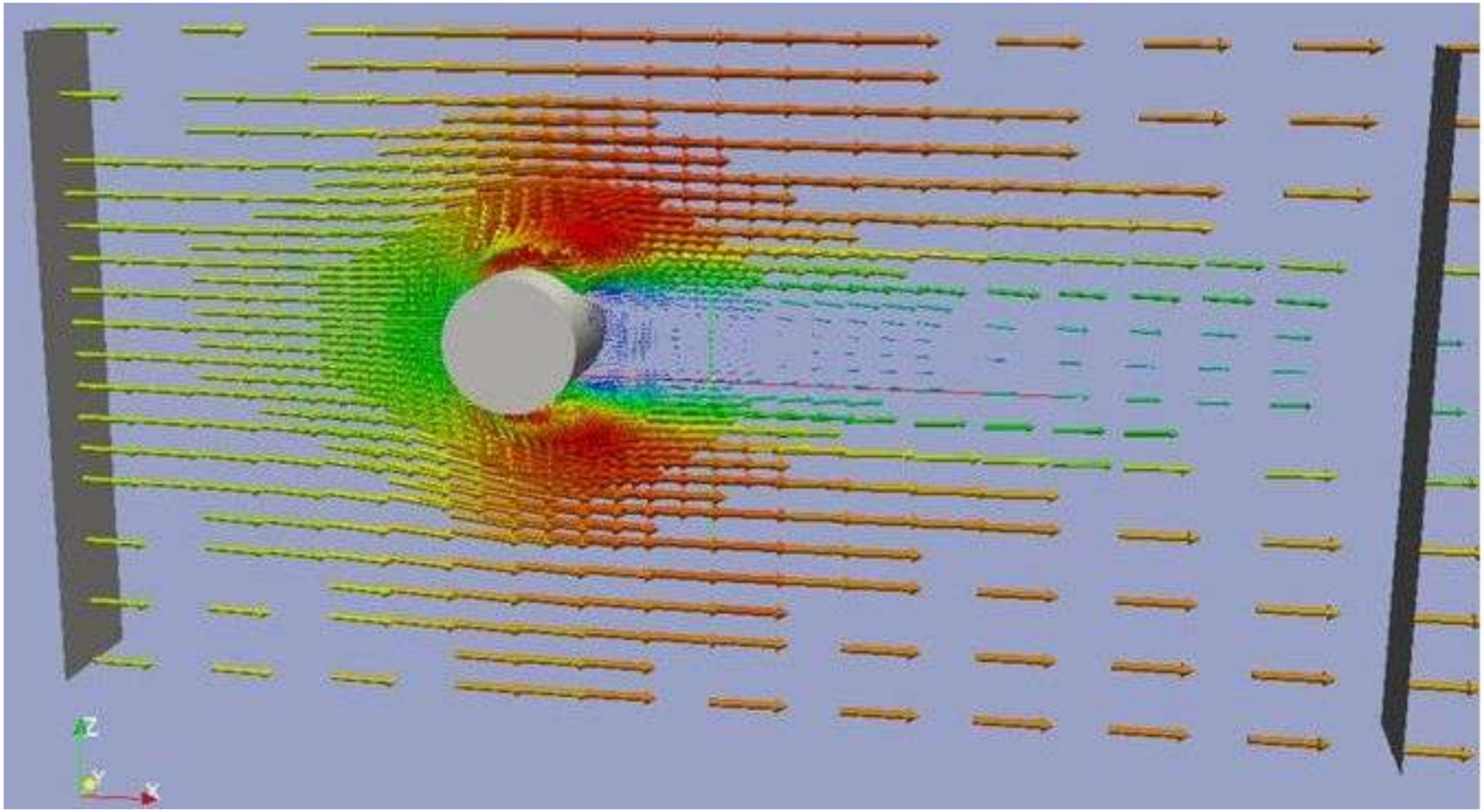
Seminar

CHAM



USP and AGG: Example #3
Computed velocity contours

The closeness of the vectors reveals the local grid



USP and AGG: Example #4; faucet for mixing hot and cold water

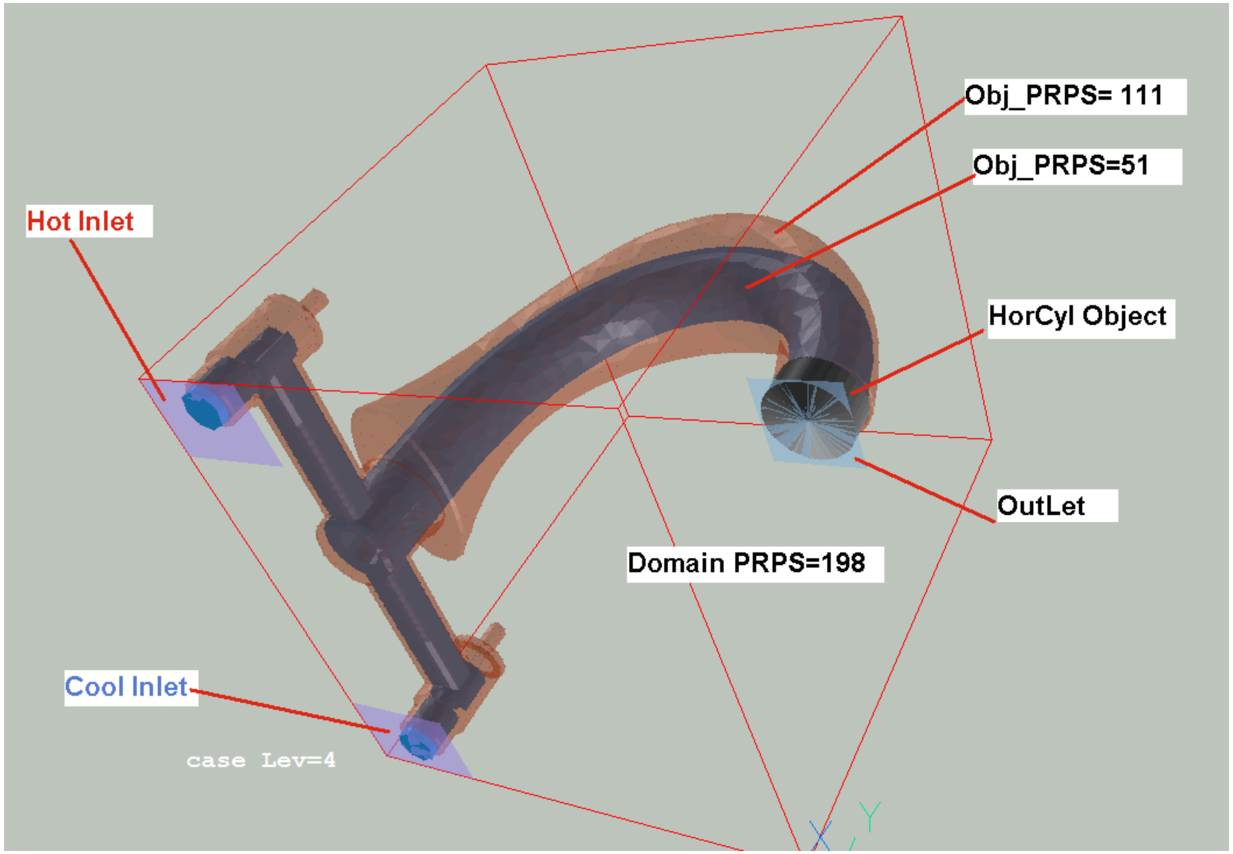
Seminar

CHAM

Structured PHOENICS could have handled example #3 well; but it would be **less efficient** if applied to example #4 .

The object represents a domestic hot-&-cold-water tap.

Only internal passages require CFD analysis; but the solid parts conduct heat.

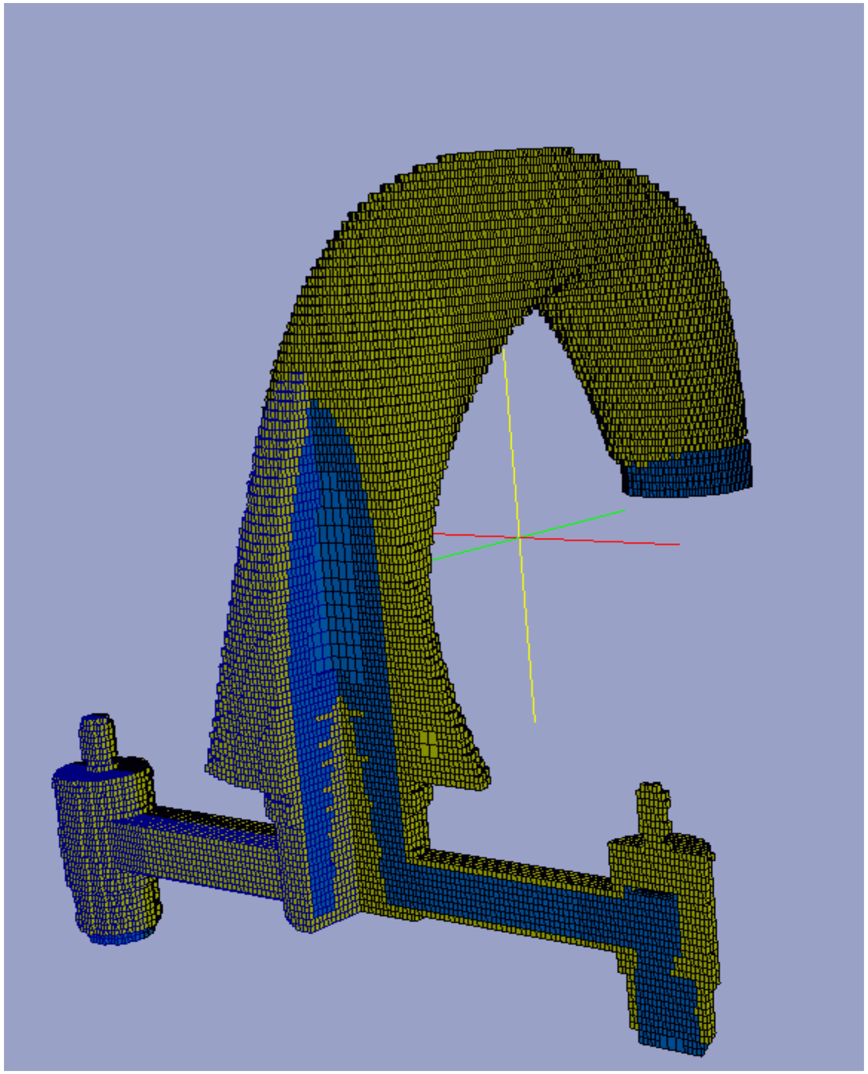


USP and AGG: Example #4
Grid and PRPS (material index) contours

MaxLevel = 4;
i.e. there are 4 levels of grid refinement.

The total number of cells is: 174 000

The fluid space is coloured blue; the solid space is coloured olive.

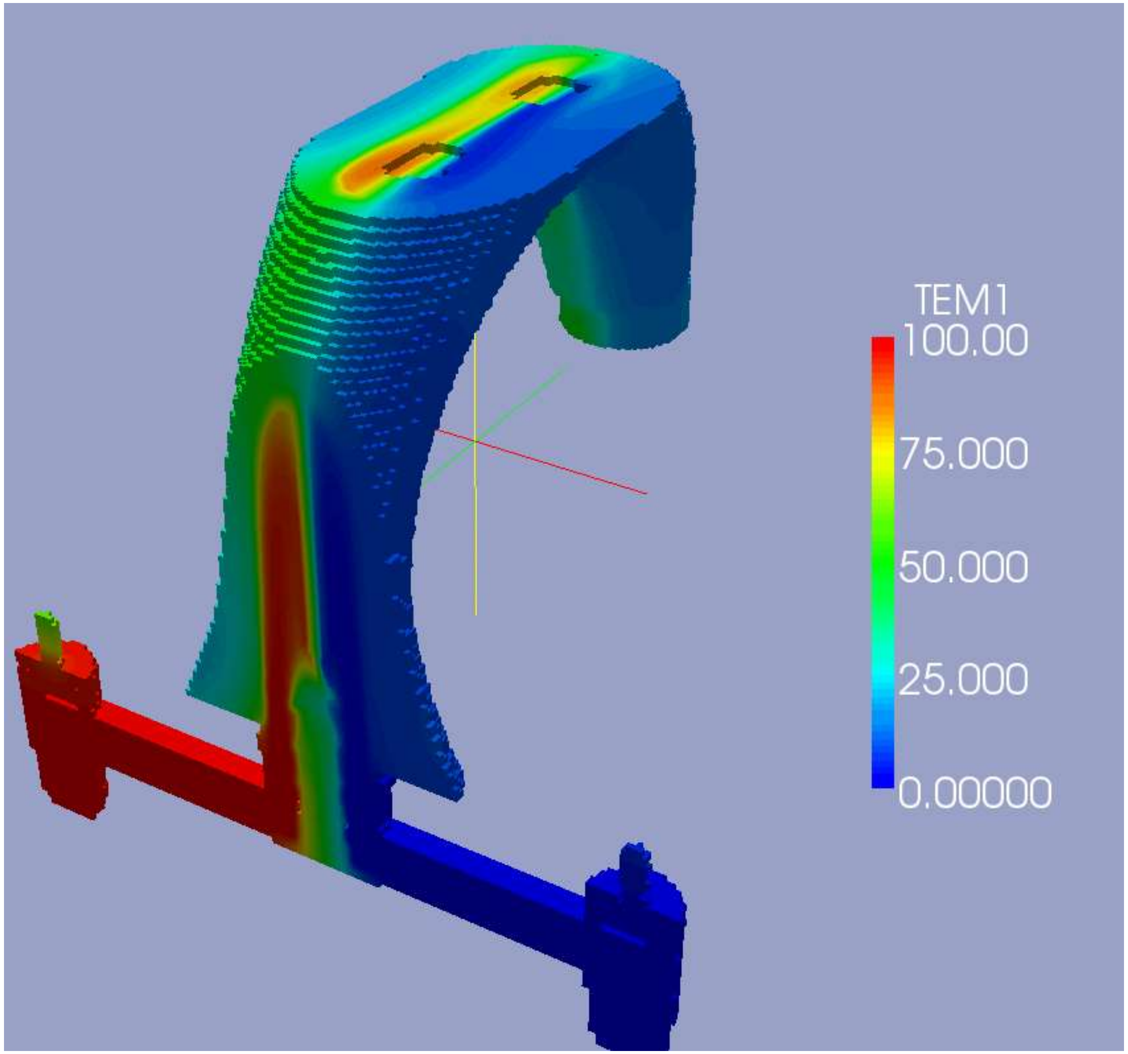


USP and AGG: Example #4
Temperature contours

The public-domain package **PARAVIEW** is here used for displaying temperature contours on:

- two cutting planes, and
- part of the outside of the faucet.

The **temperature range** is from **0** to **100** degrees.

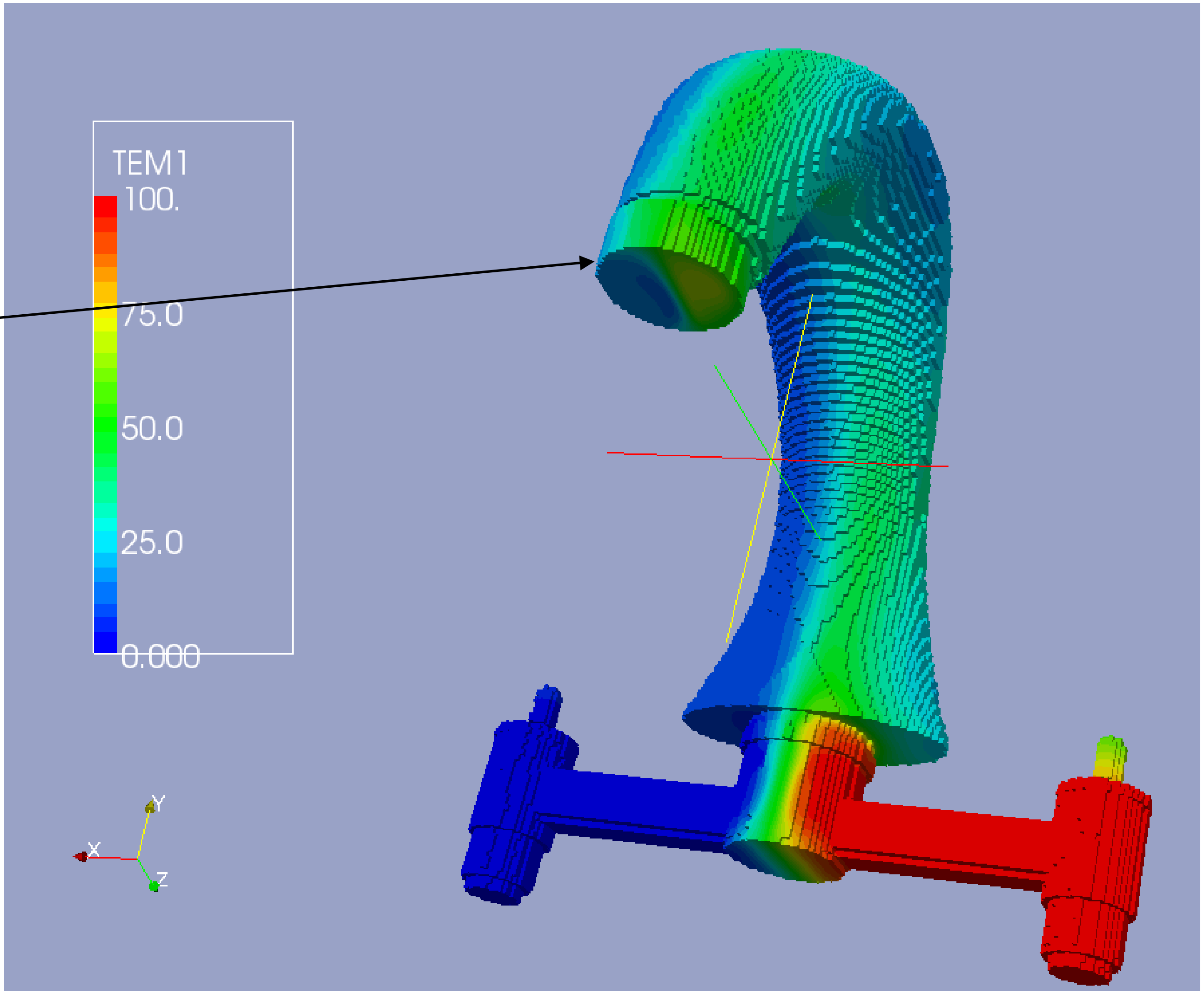


USP and AGG: Example #4; surface-temperature contours

Seminar

CHAM

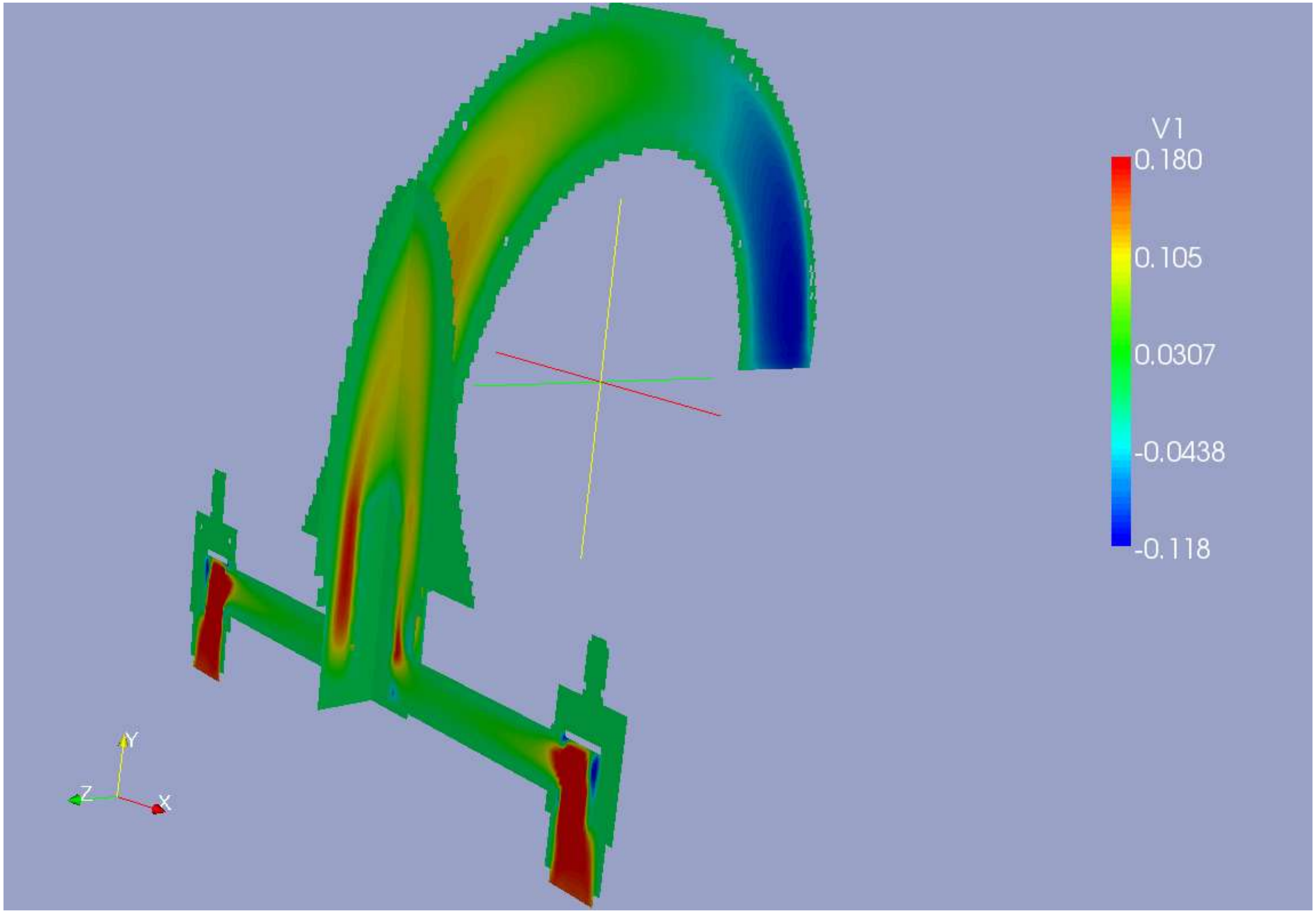
A fictitious cylindrical object has been attached to the outlet so as to enable the outlet pressure to be specified



USP and AGG: Example #4; Vertical velocity contours

Seminar

CHAM



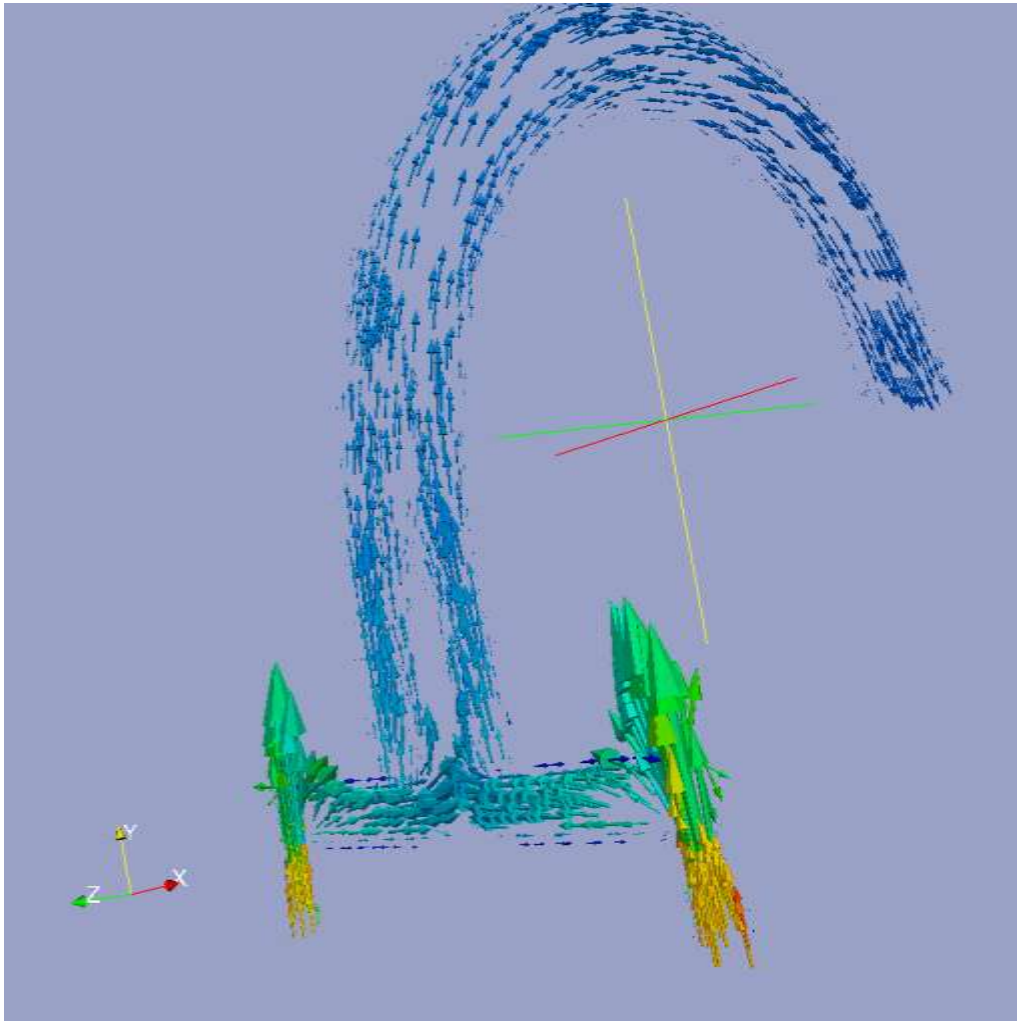
USP and AGG: Example #4; Velocity vectors (coloured by pressure)

Seminar

CHAM

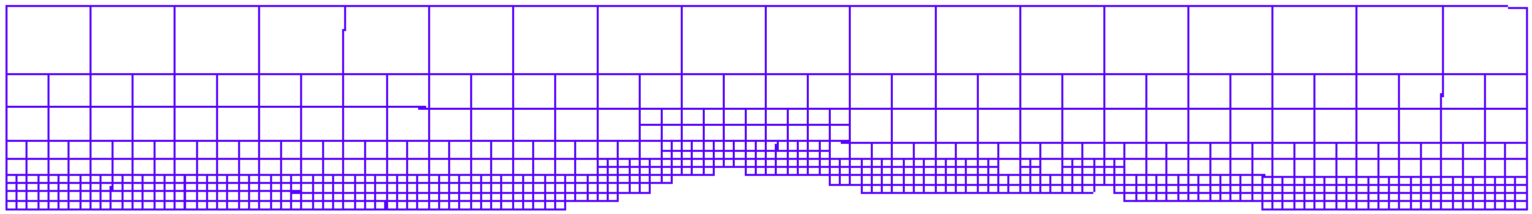
The arrows show the hot and cold entering streams, which flow towards each other.

They then join and flow out together along the curved tube to the outlet.



PHOENICS Unstructured
Flow over terrain

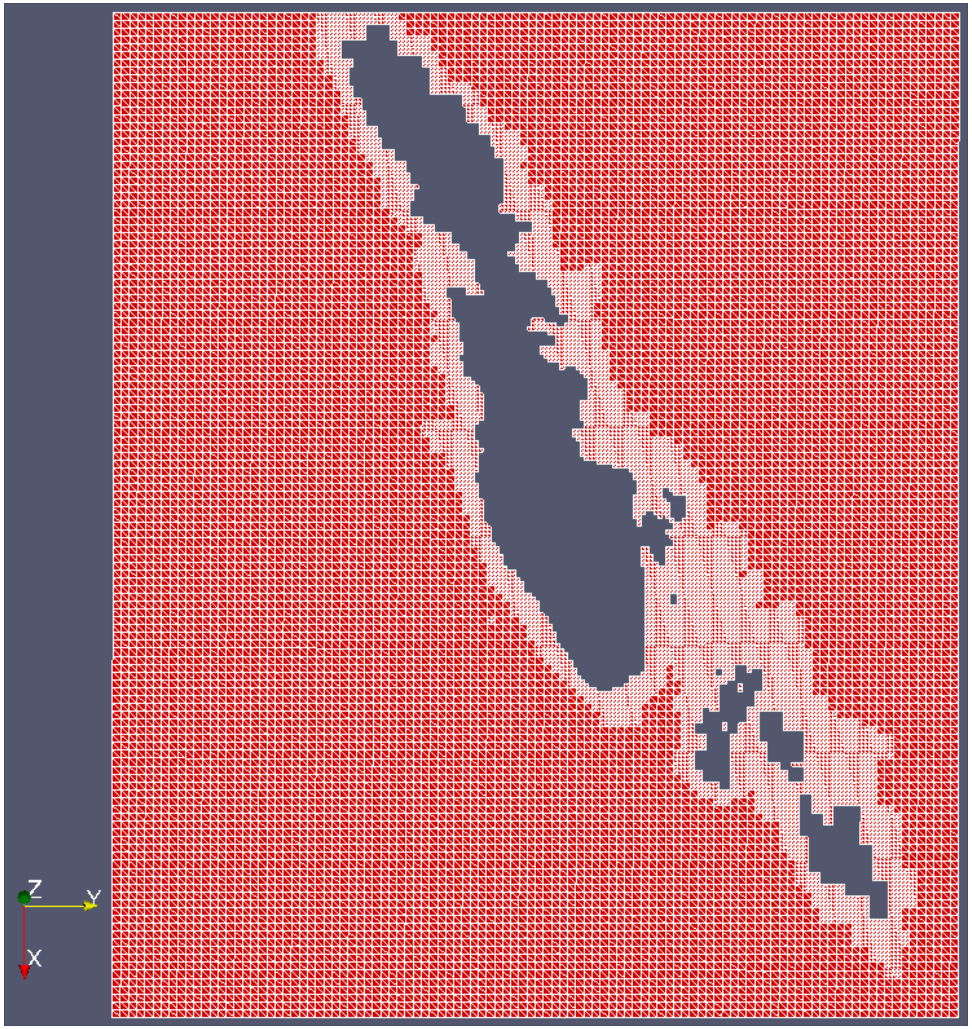
USP is particularly useful for **flow-over-terrain** problems where **fine** grids are required **near the ground,** whereas **coarser** ones suffice for **higher altitudes**.



PHOENICS Unstructured
USP grid

Seminar

CHAM



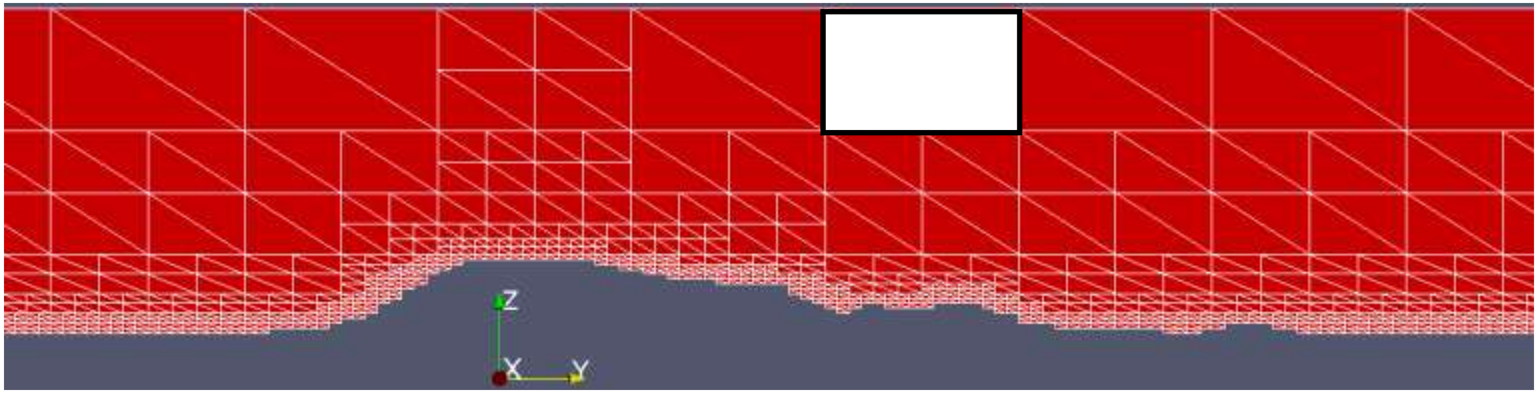
The USP grid for the given terrain at Z = 75 m

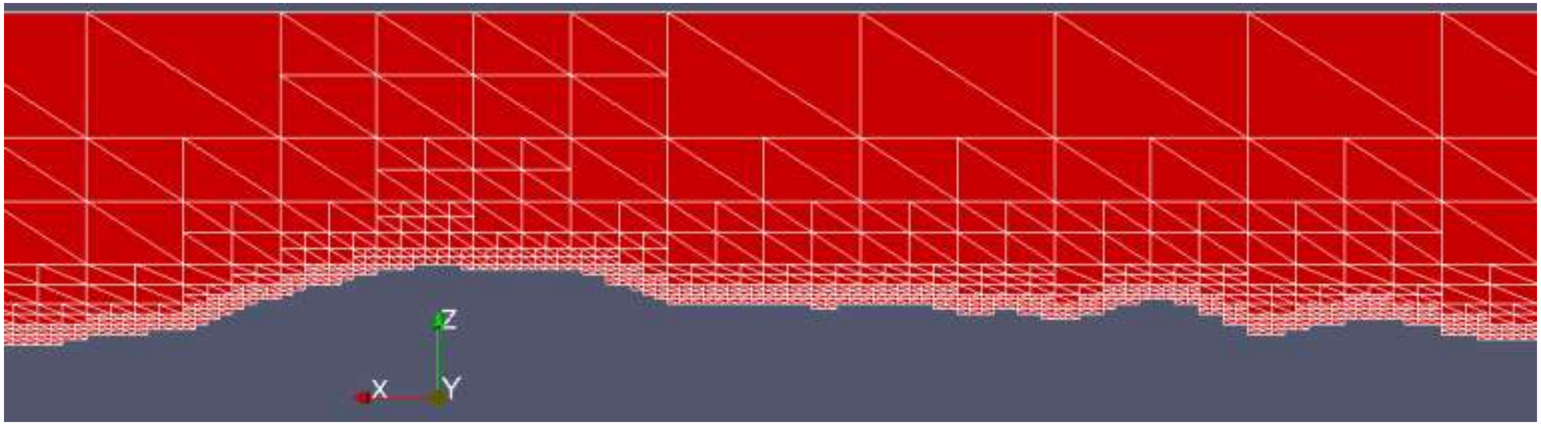
PHOENICS Unstructured
USP grid

Seminar

CHAM

4 levels of refinement were used for the generation of this grid. The white cell shows the initial cell size of the starting coarse mesh.

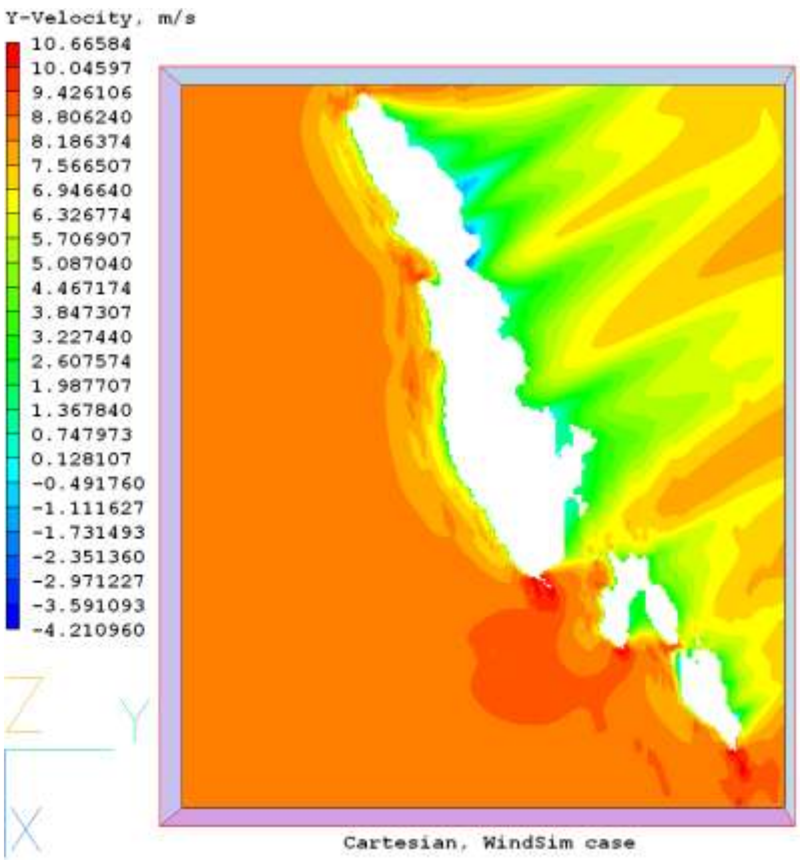




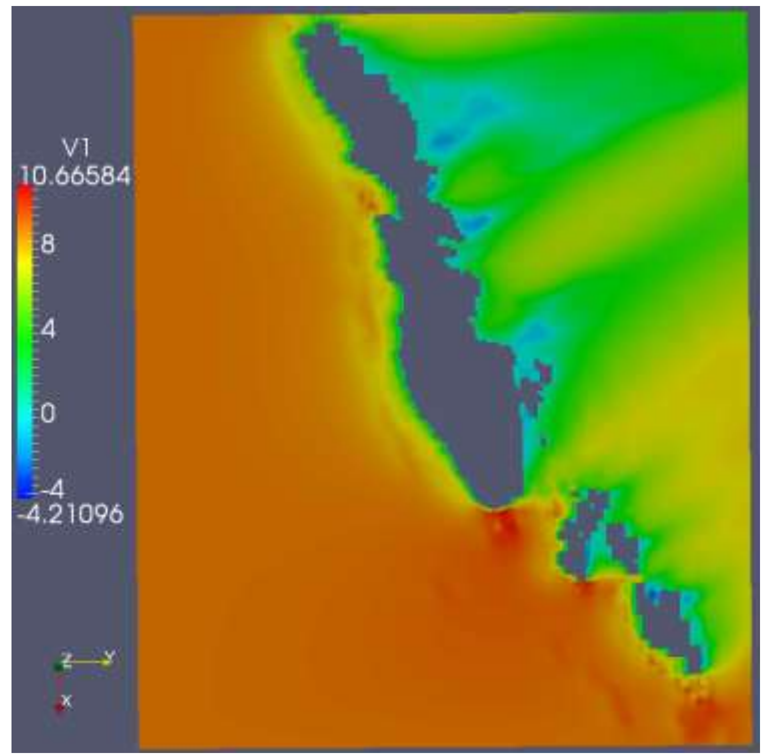
PHOENICS Unstructured
SP vs USP

Seminar

CHAM



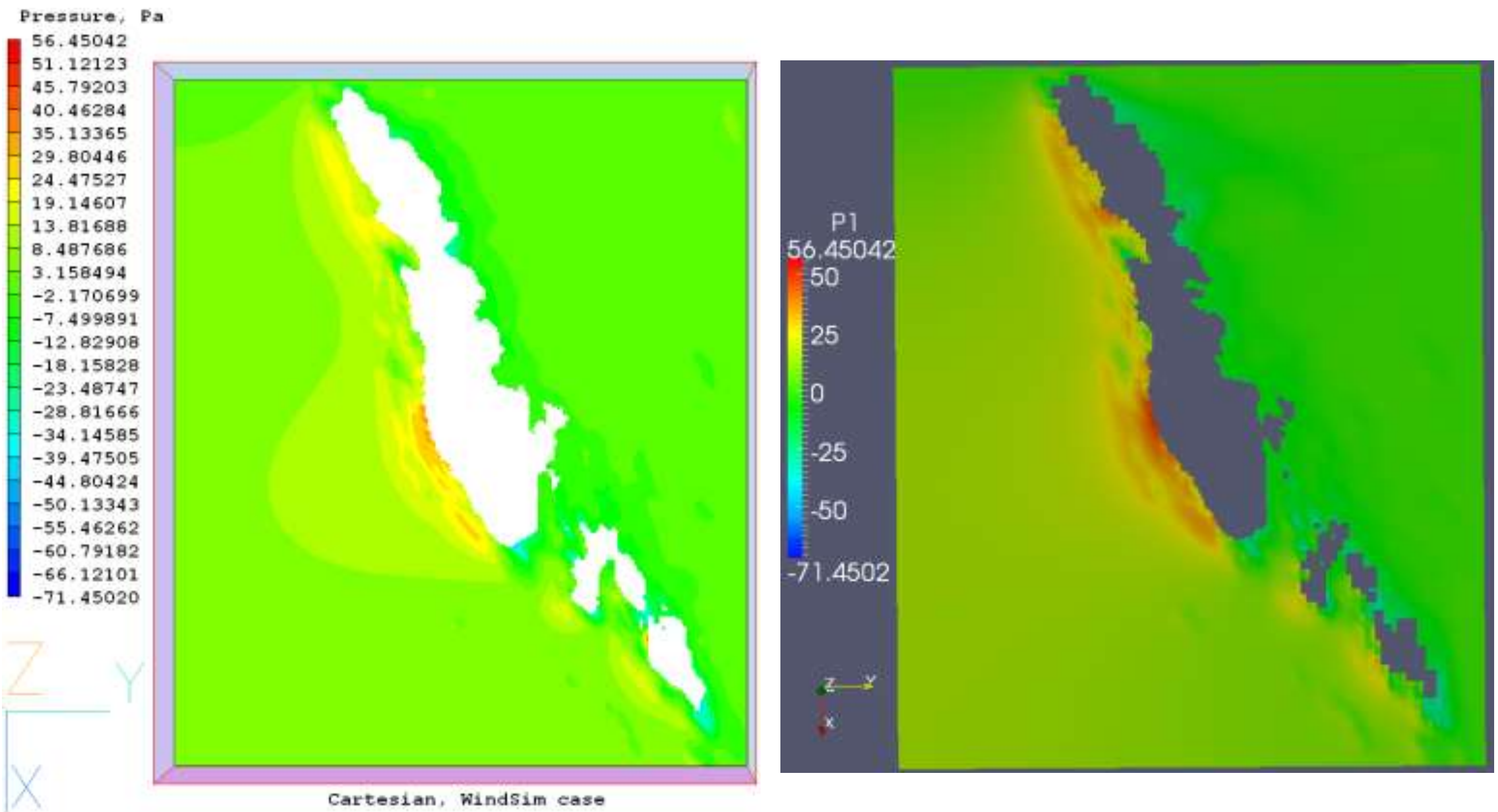
Cartesian, WindSim case



PHOENICS Unstructured
SP vs USP

Seminar

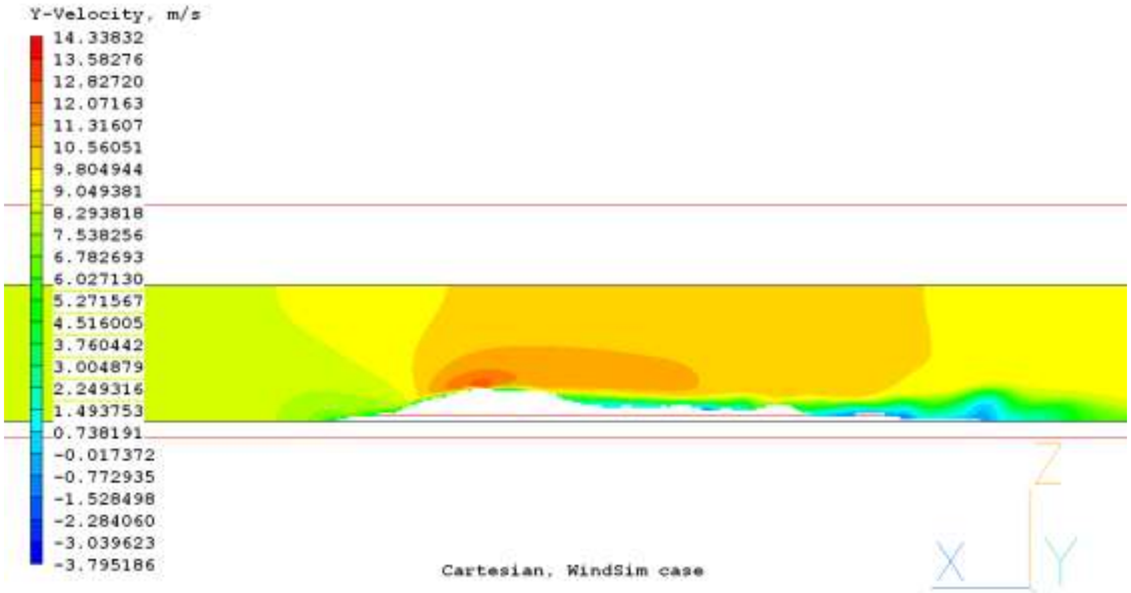
CHAM

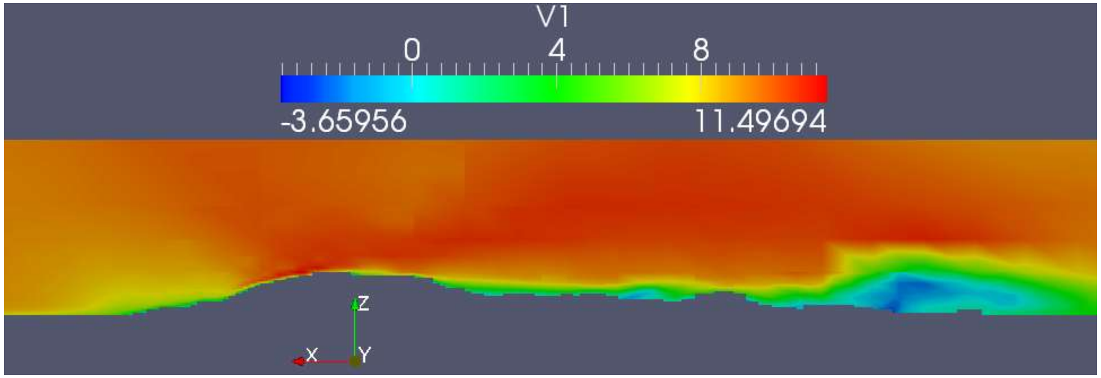


PHOENICS Unstructured
SP vs USP

Seminar

CHAM

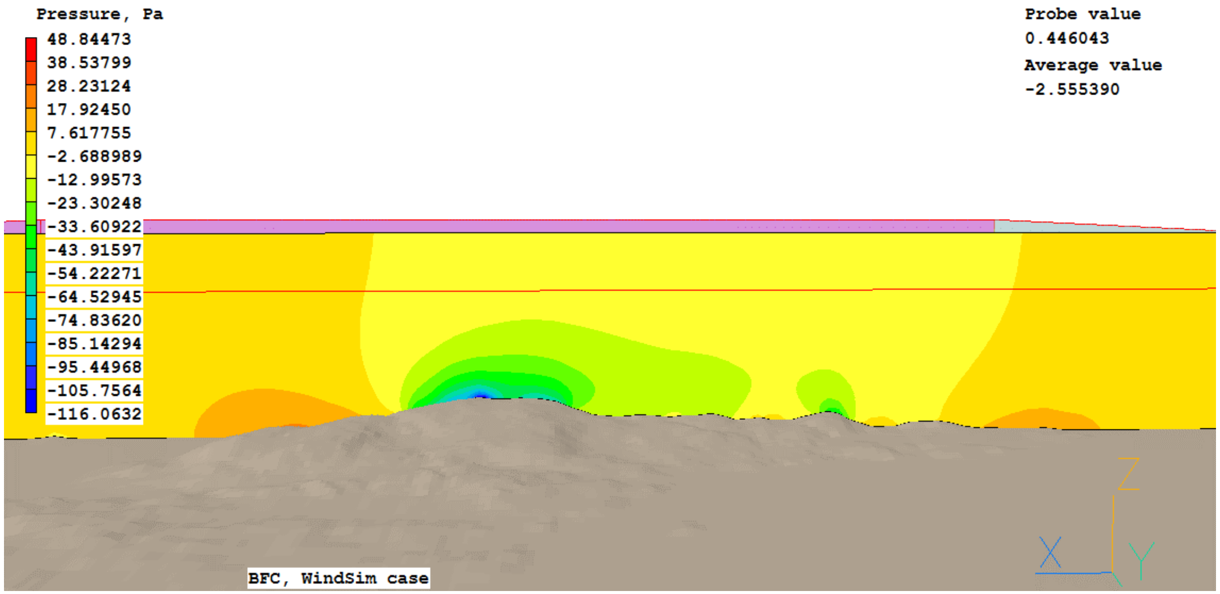


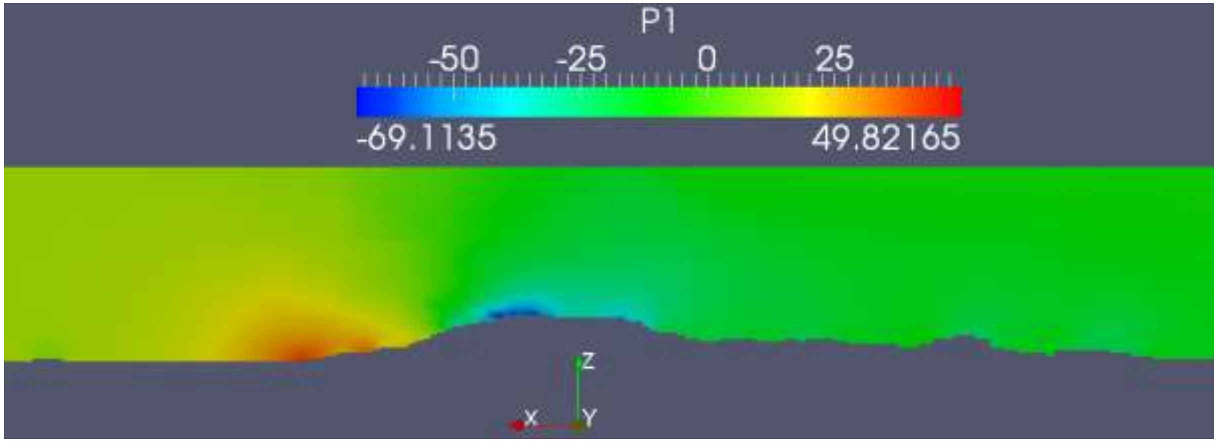


PHOENICS Unstructured
BFC vs USP

Seminar

CHAM

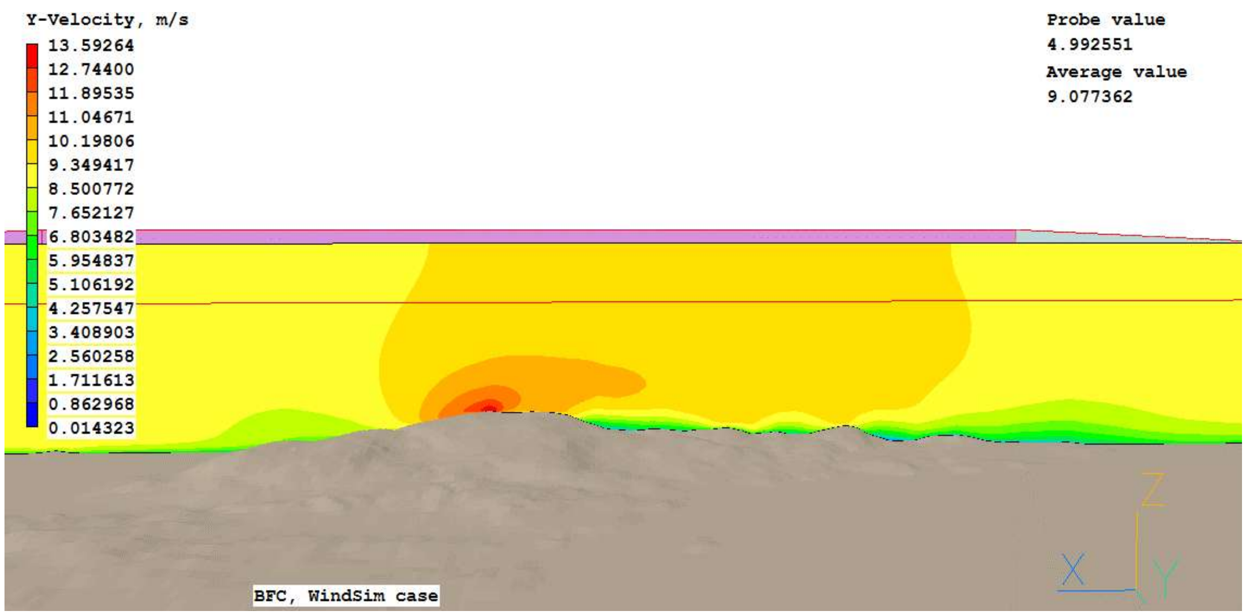


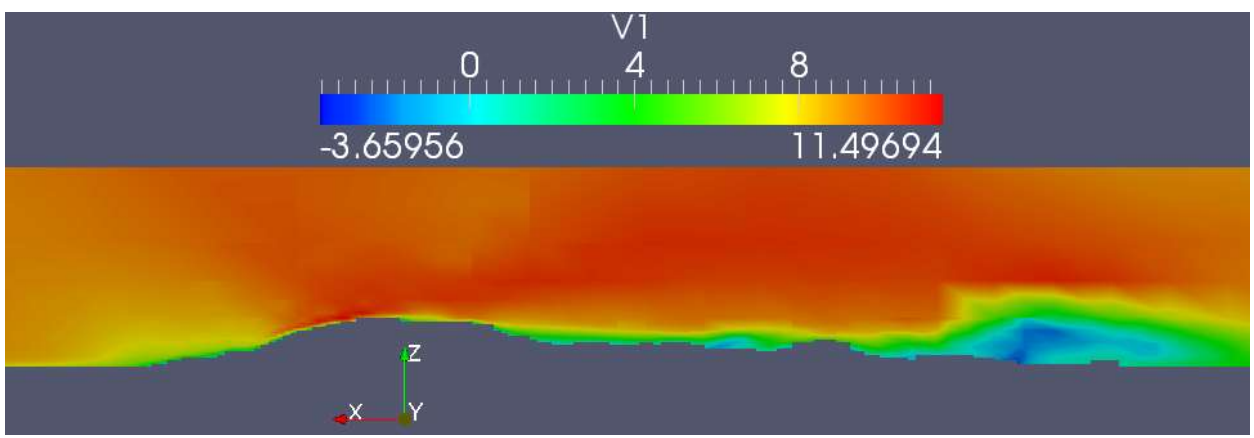


PHOENICS Unstructured BFC vs USP

Seminar

CHAM

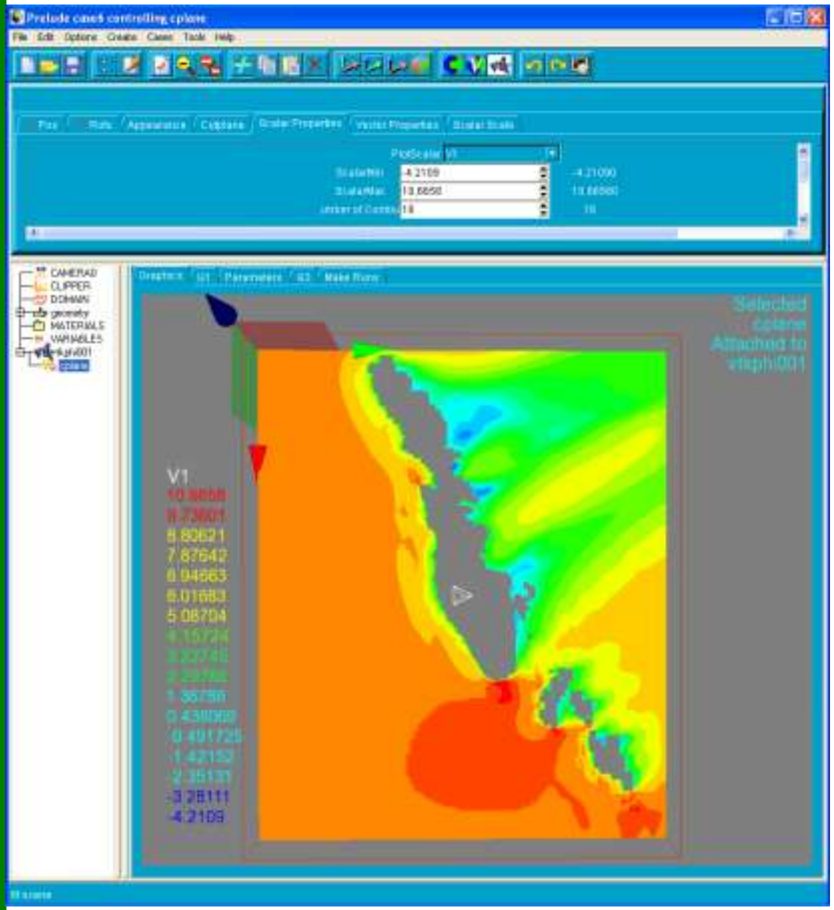


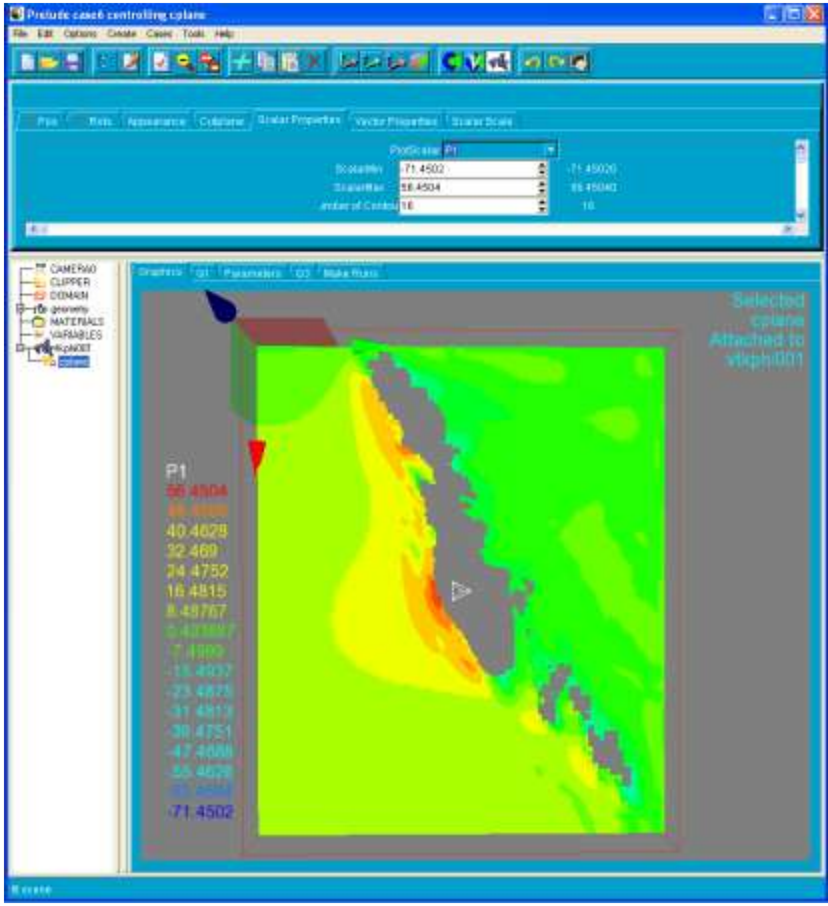


Display of USP results with PRELUDE

Seminar

CHAM





In addition to Paraview, CHAM's new pre -(and post-) processor PRELUDE can now display USP results.

Comparison of SP, BFC and USP for terrain-type problems

The computed results of Structured PHOENICS (**SP**), PHOENICS using Body-Fitted Coordinates (**BFC**) and Unstructured PHOENICS (**USP**) agree in all important respects.

Seminar

CHAM

PHOENICS

Impeller Pump Example

Impeller Pump Example

- This presentation outlines the modeling of a simple impeller pump using PHOENICS.
- The model shown is a transient, moving body analysis, however steady state and start-up simulations can also be run with PHOENICS.
- The work was part of a demonstration for Armfield Ltd

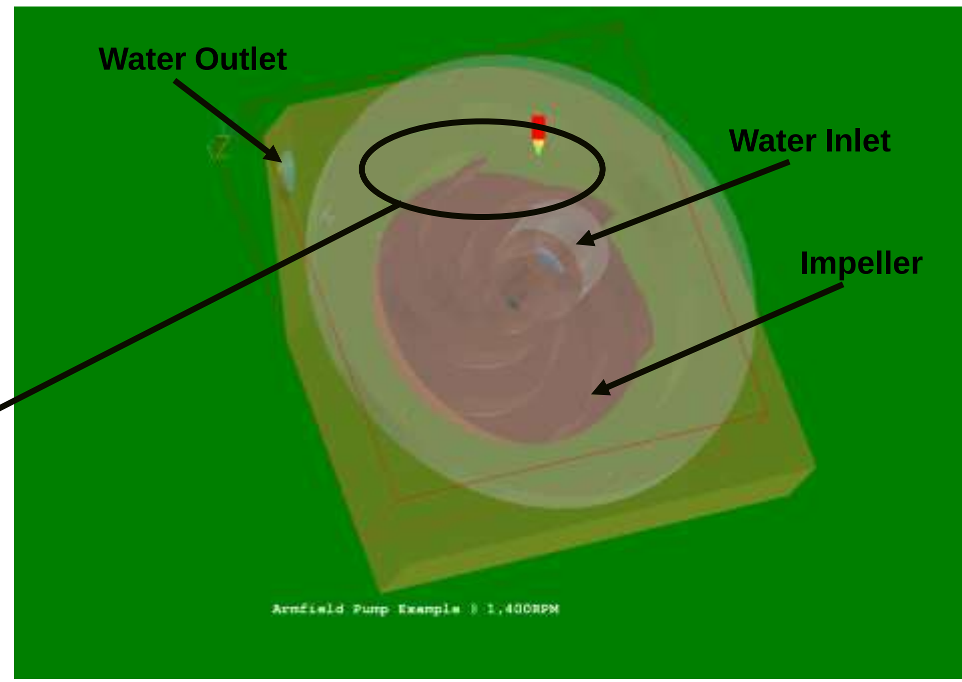
The Geometry

Seminar

CHAM

General and detailed views of the pump geometry in the PHOENICS VR viewer.

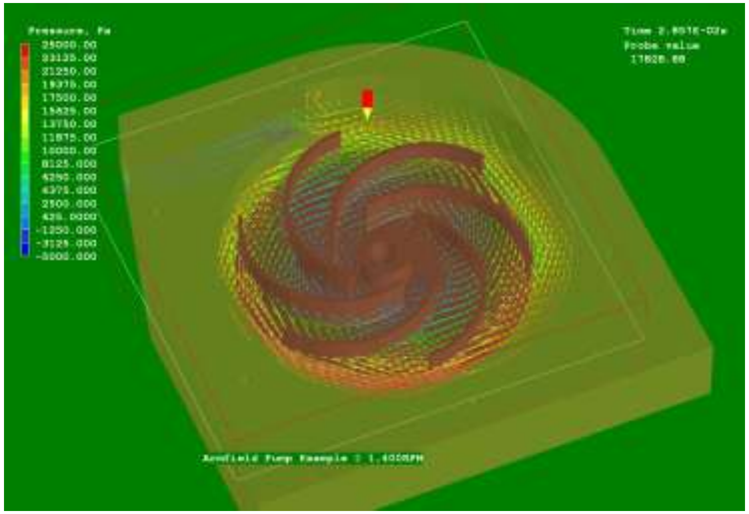
- The cover is shown as transparent to aid visualisation.
- In this case the geometry was from stl files. Most CAD systems are capable of generating this file format.



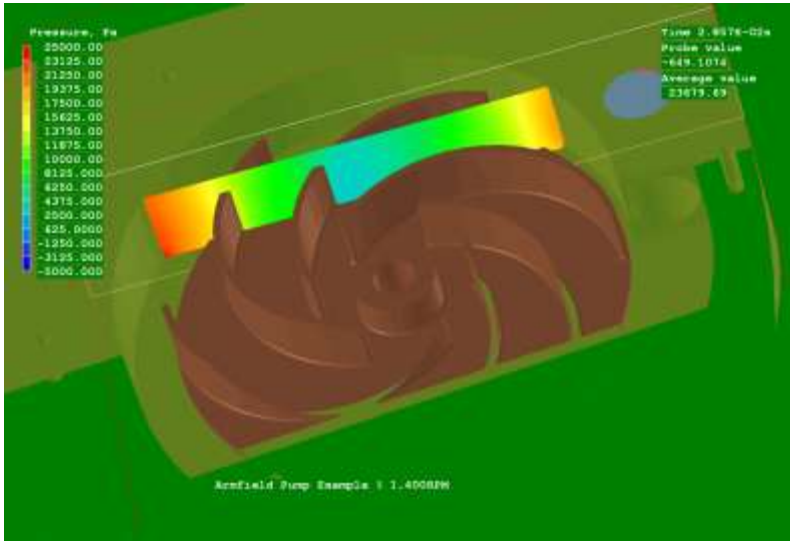
Results - Pressure

Seminar

CHAM



Sectional pressure profile and vectors through and around the impeller.

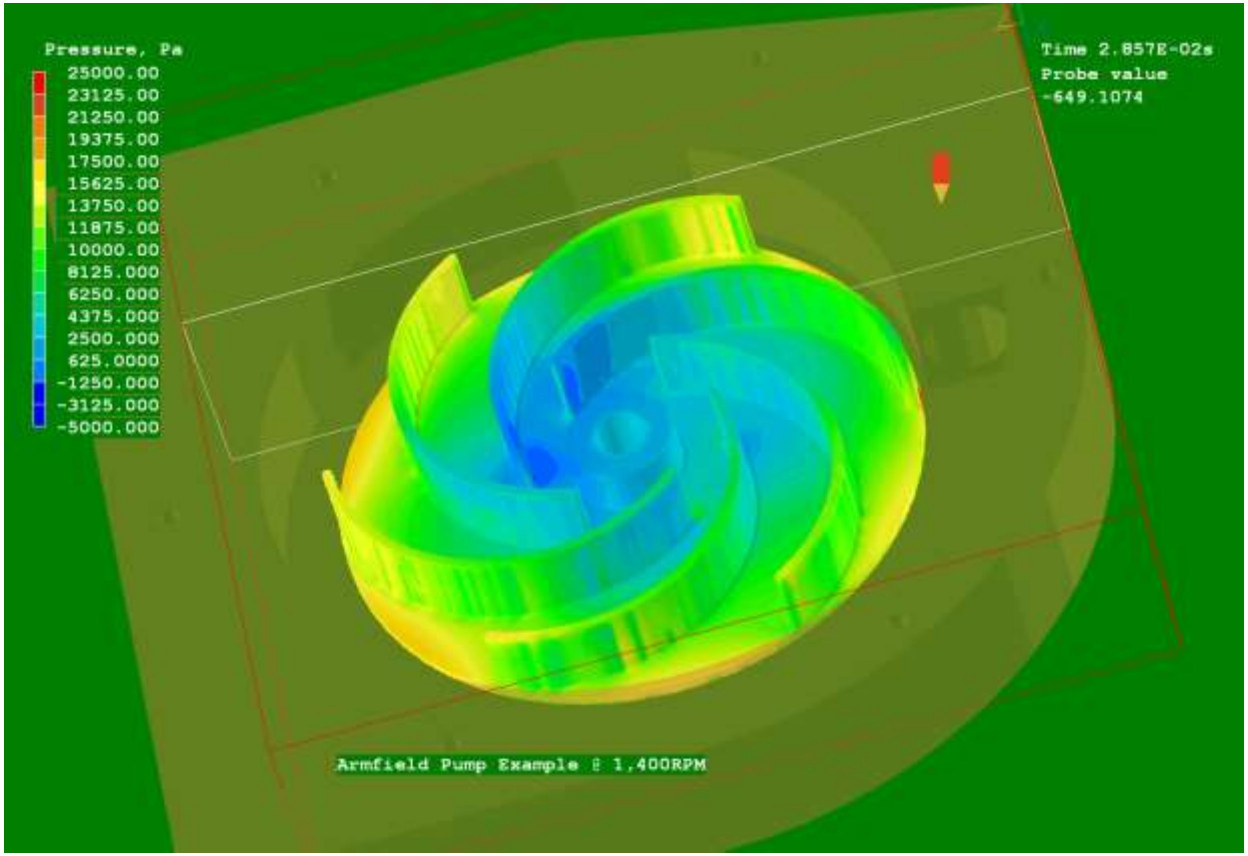


Results - Pressure

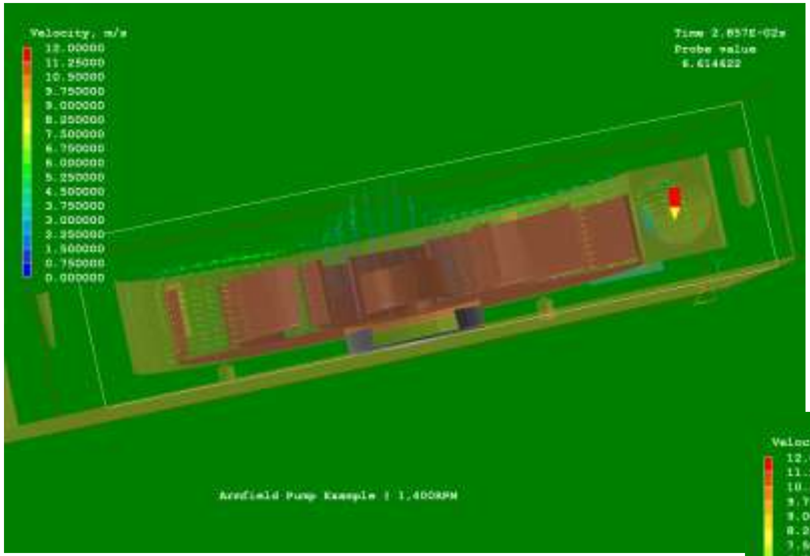
Seminar

CHAM

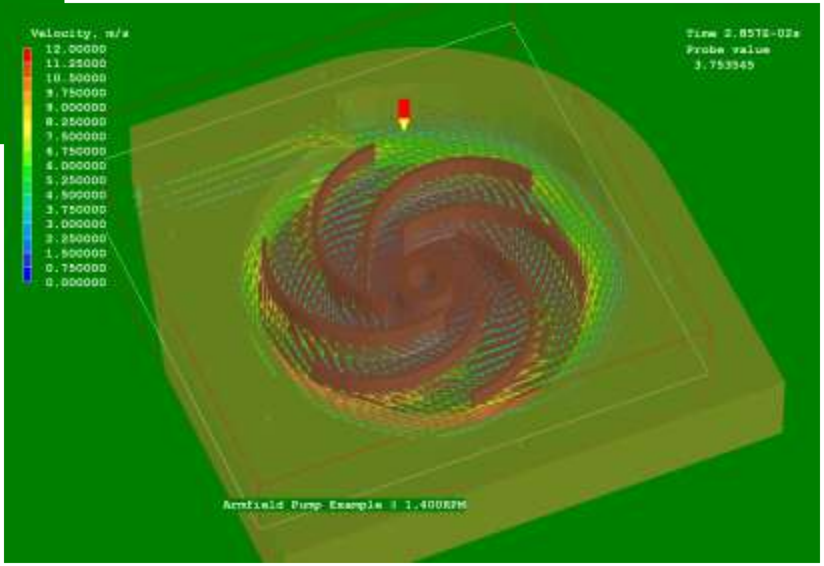
Surface pressure profile on the impeller.



Results - Velocity



Velocity vectors within the pump, showing use of the 'near-plane' function to view inside the impeller cross section.

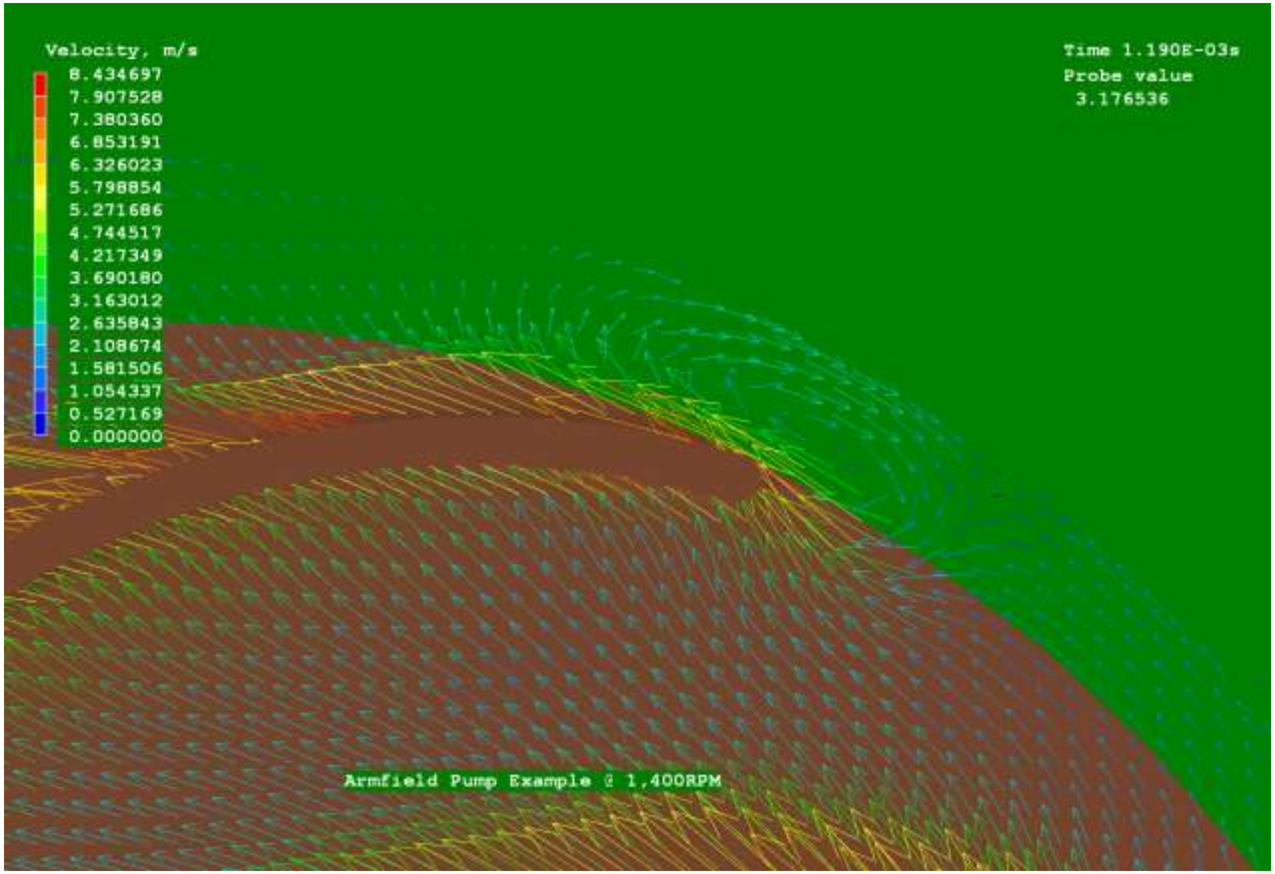


CHAM

Results - Velocity

CHAM

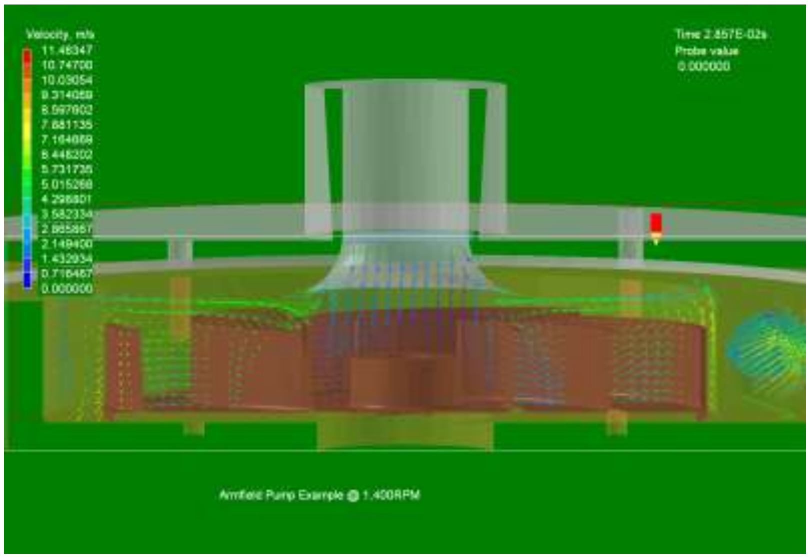
Detailed velocity vectors showing turbulence at the impeller tip.



Results - Velocity

Seminar

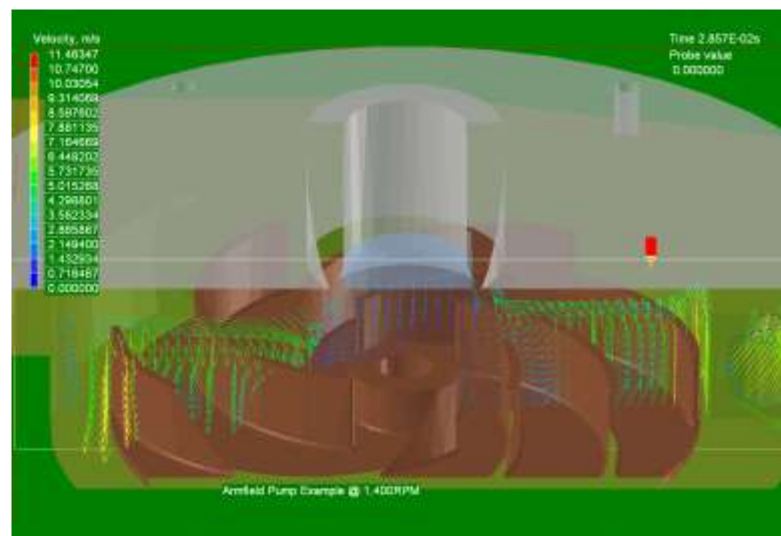
CHAM



From the geometry supplied the diameter of the outlet appears to be 18mm diameter. Therefore volume flow rate is given by…

Area of outlet = 2.54 E-4 m
Average velocity at outlet = 4.78m/s

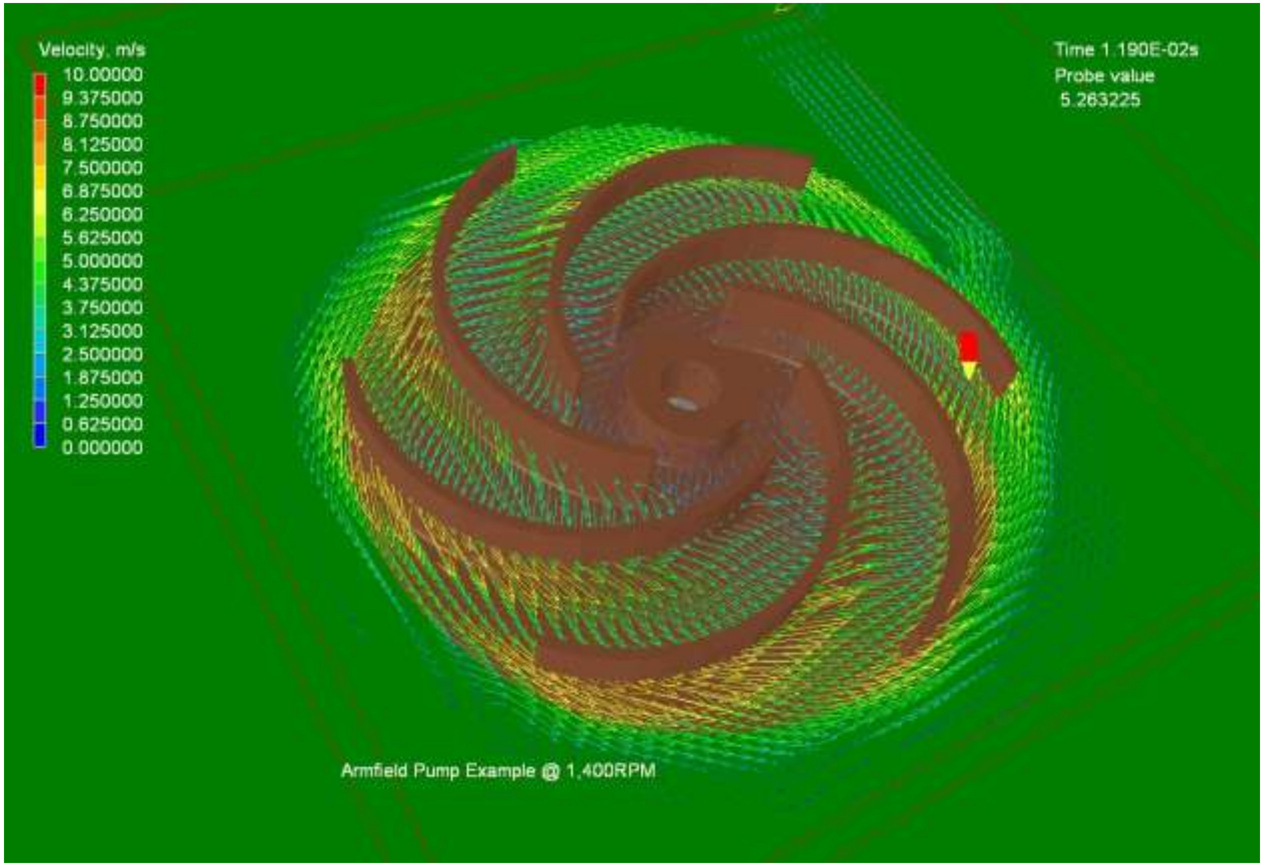
Volume flow rate = 0.00122m3/s
Experimental flow rate = 0.001m3/s



Results - Animation

Seminar

CHAM



CHAM

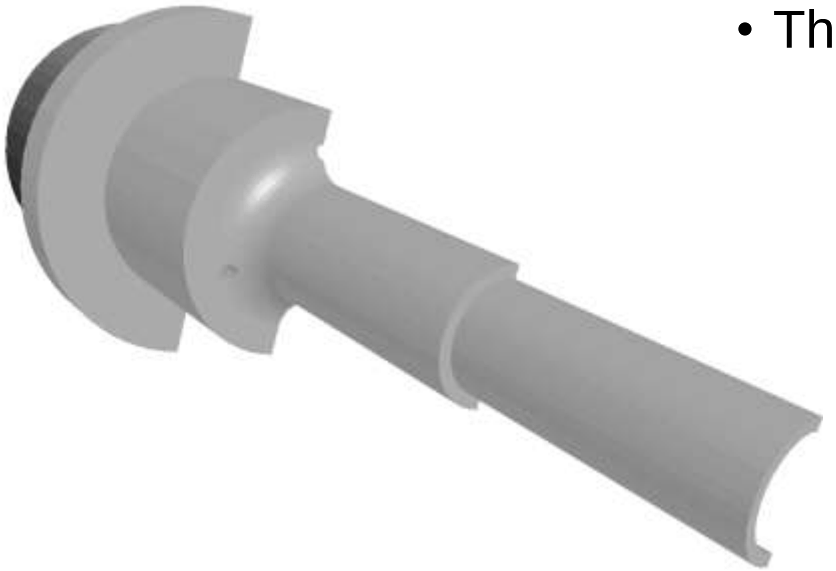
PHOENICS
Thermocouple Example

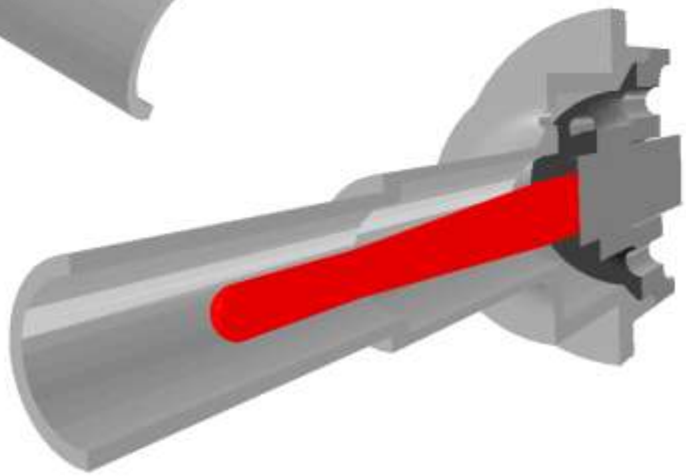
Thermocouple

- PHOENICS was used to calculate the flow and temperature distribution inside a thermocouple.
- The thermocouple is used to measure the temperature in a jet-engine combustion chamber.
- The operating conditions are:
  - pressure around 3bar
  - Temperature around 950ºC
- Calculations were performed for a 2D section, and also for a 3D 180º section.

Thermocouple - Geometry

- Thermocouple geometry





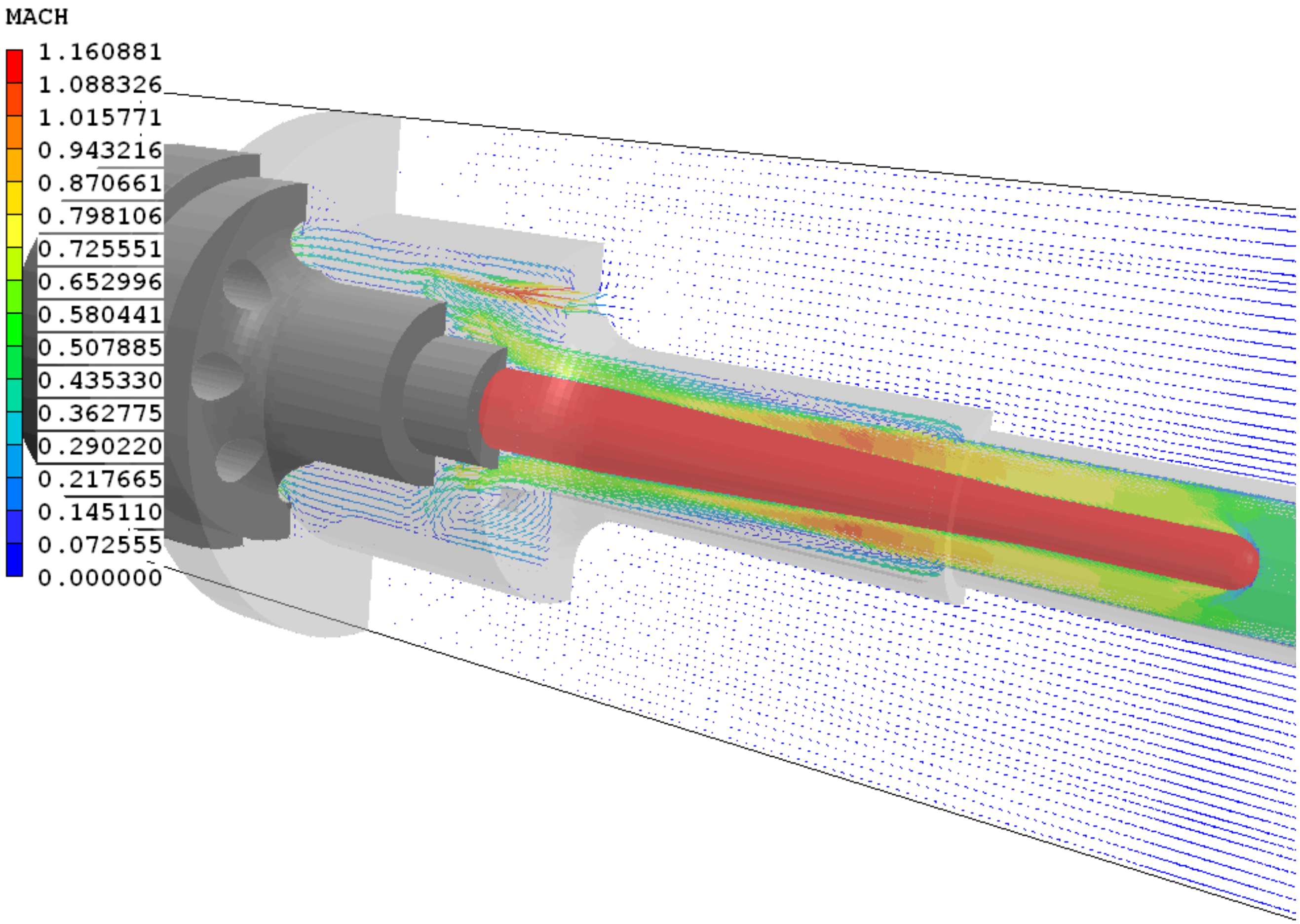
- The red part is the temperature sensor

Thermocouple - Results

Seminar

CHAM

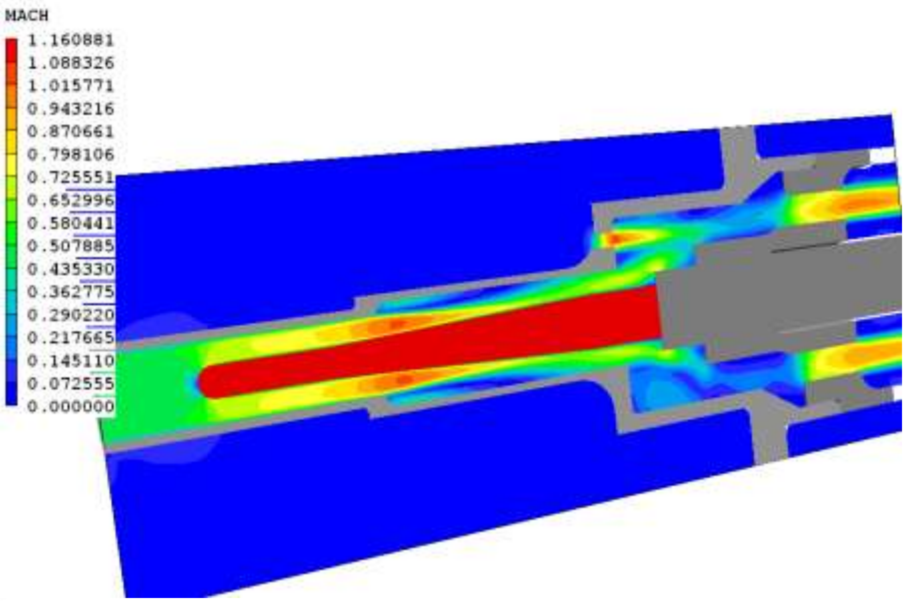
- Vectors

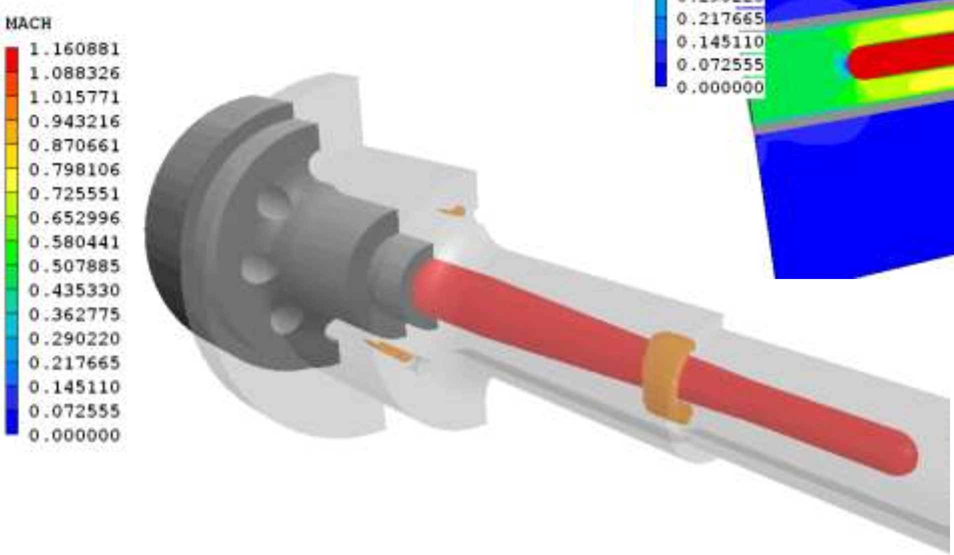


Thermocouple - Results

CHAM

- Mach Number contours

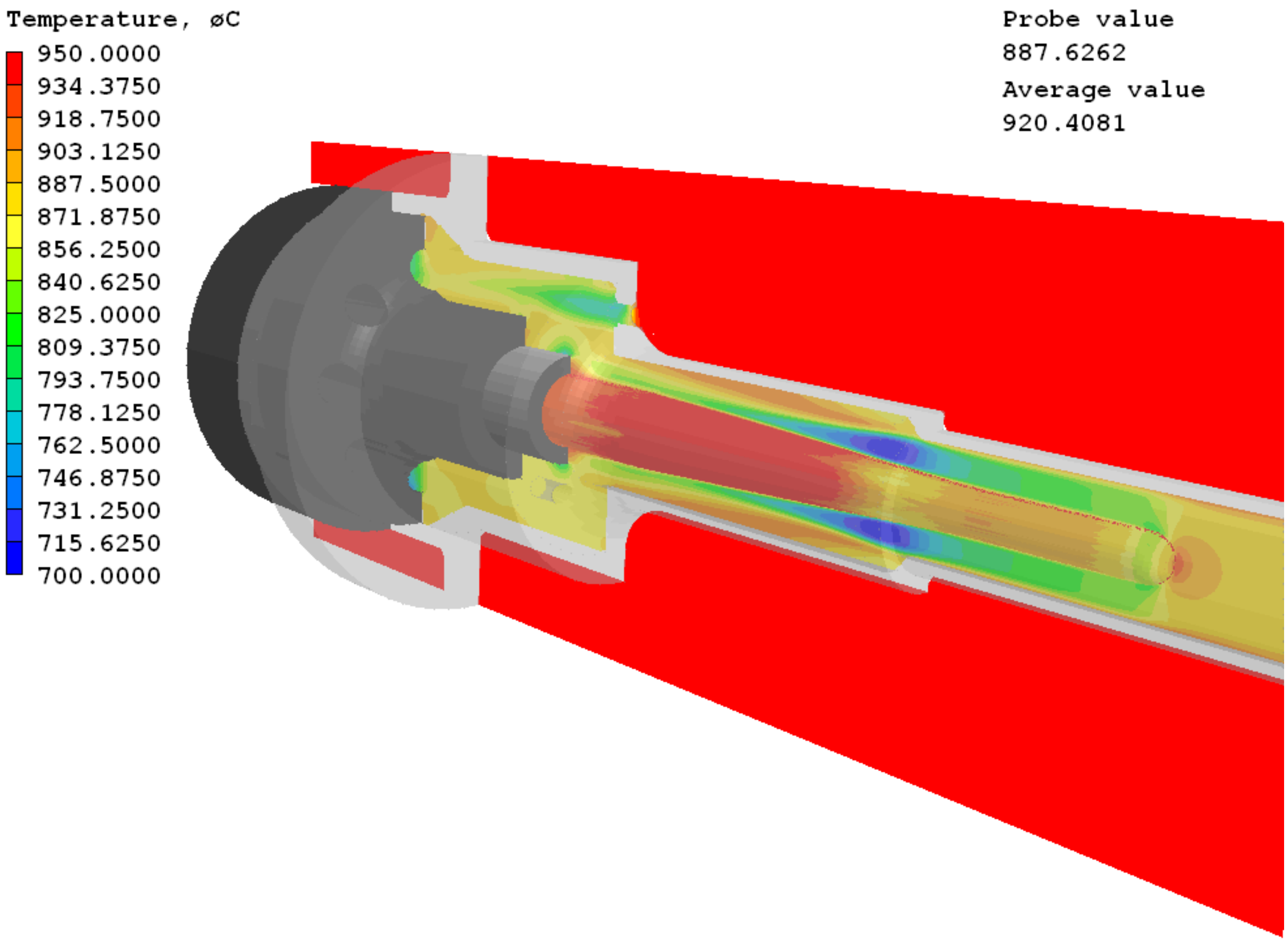




- Surface of Mach Number 1

Thermocouple - Results

- Temperature distribution

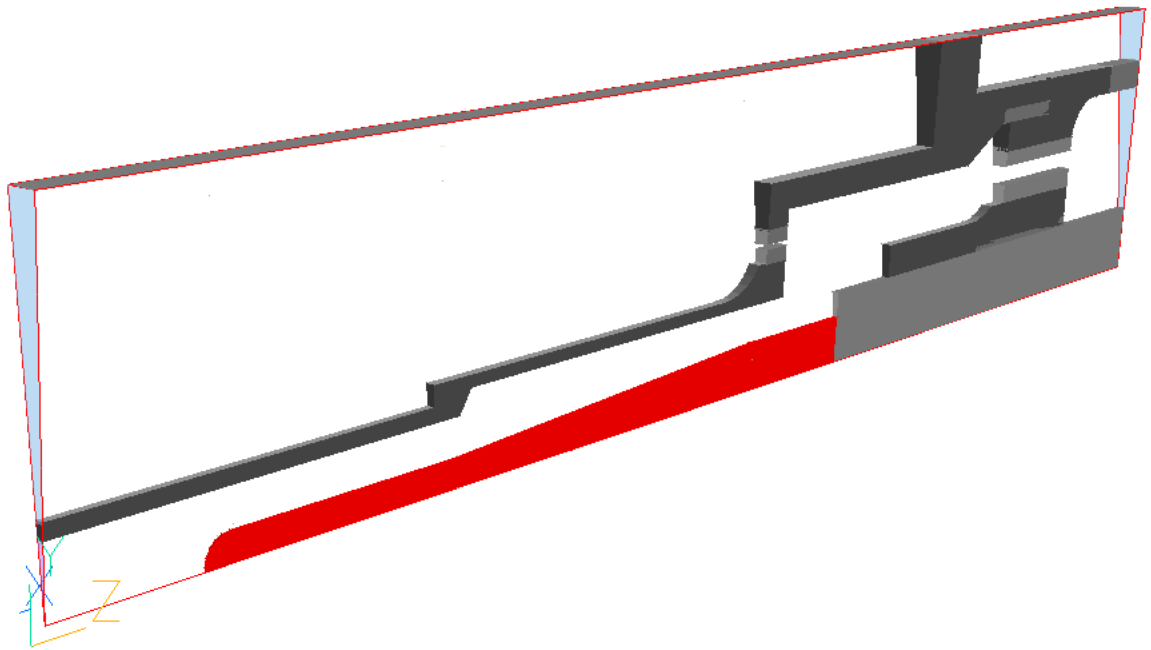


Thermocouple - Results

Seminar

CHAM

- 2D model geometry

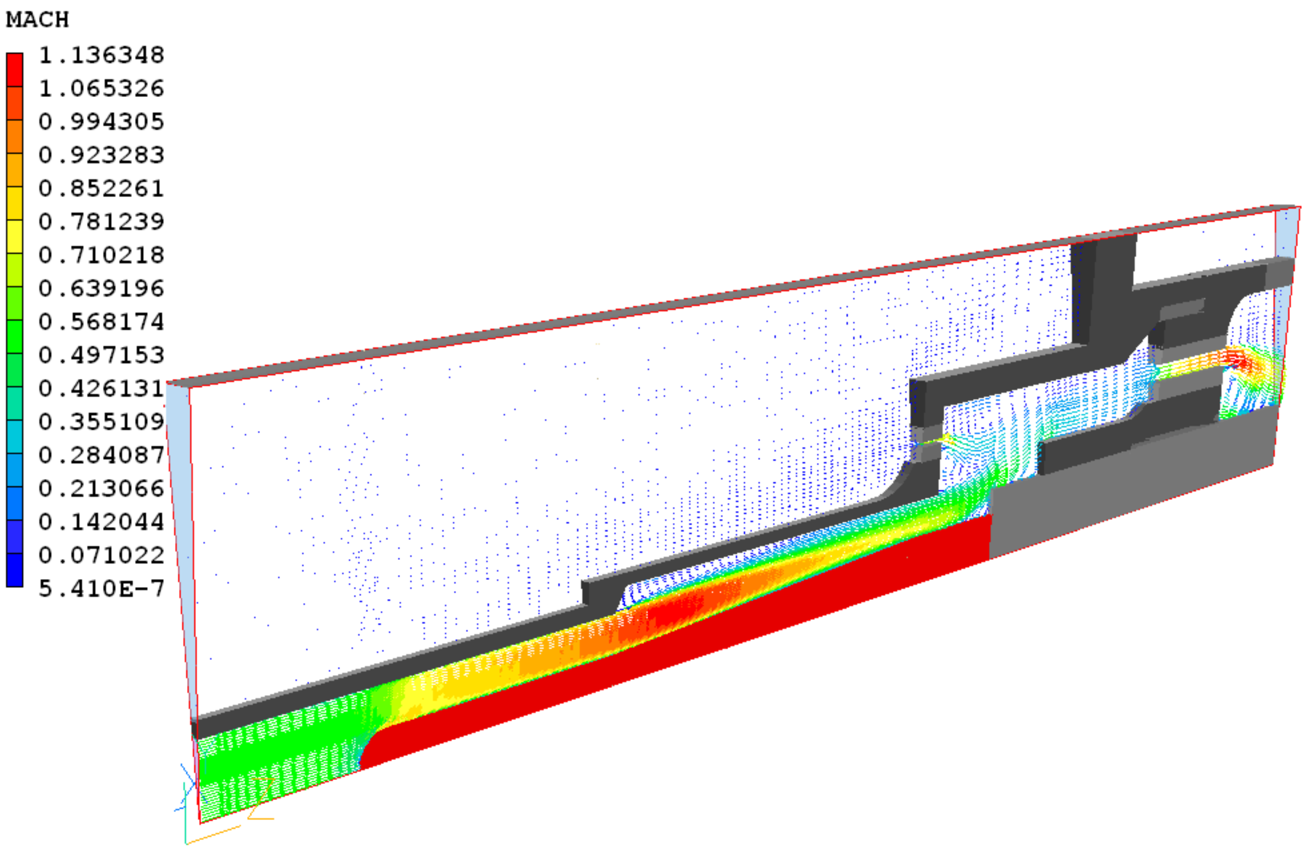


Thermocouple - Results

Seminar

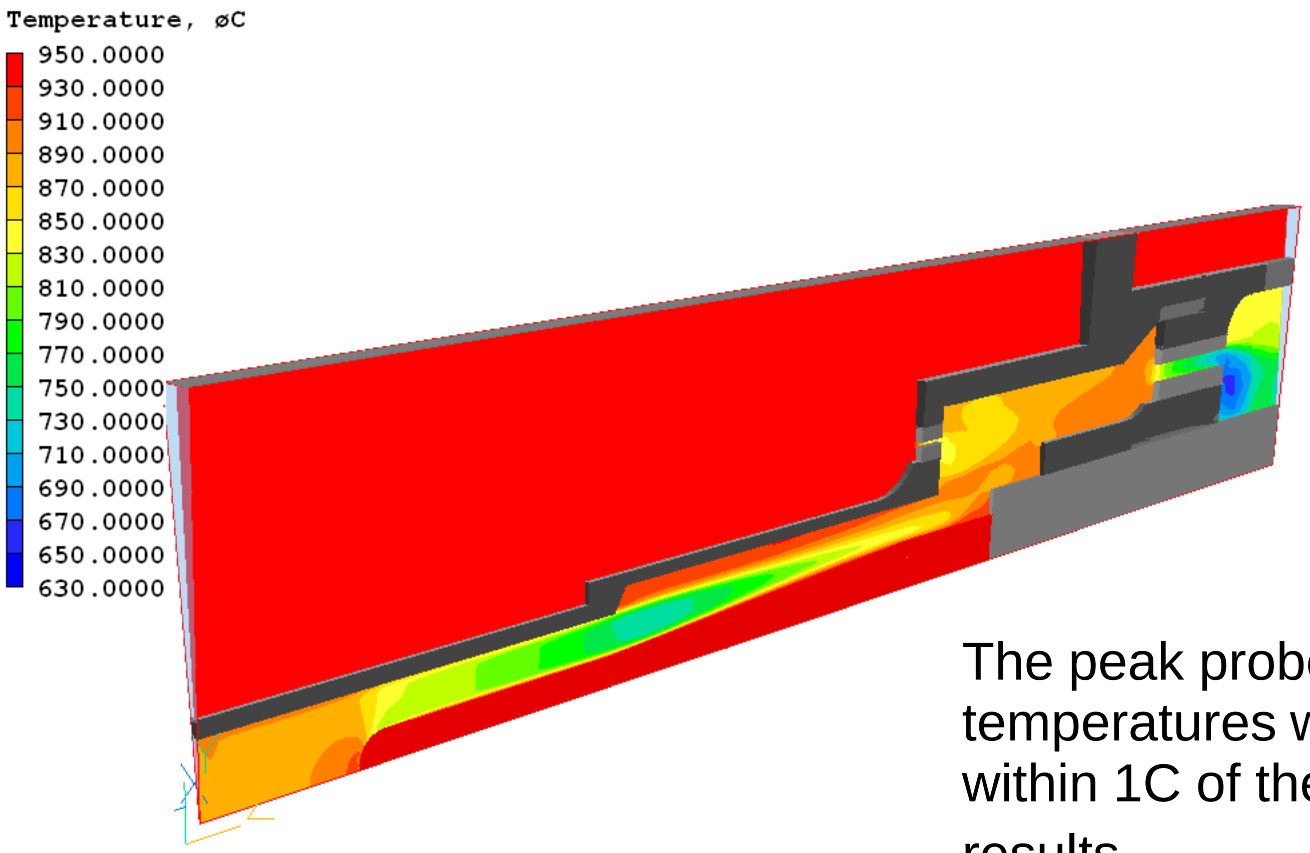
CHAM

- Vectors



Thermocouple - Results

- Temperature contours



The peak probe temperatures were within 1C of the 3D results

CHAM

PHOENICS

Vapour Extraction Example

Vapour Extraction

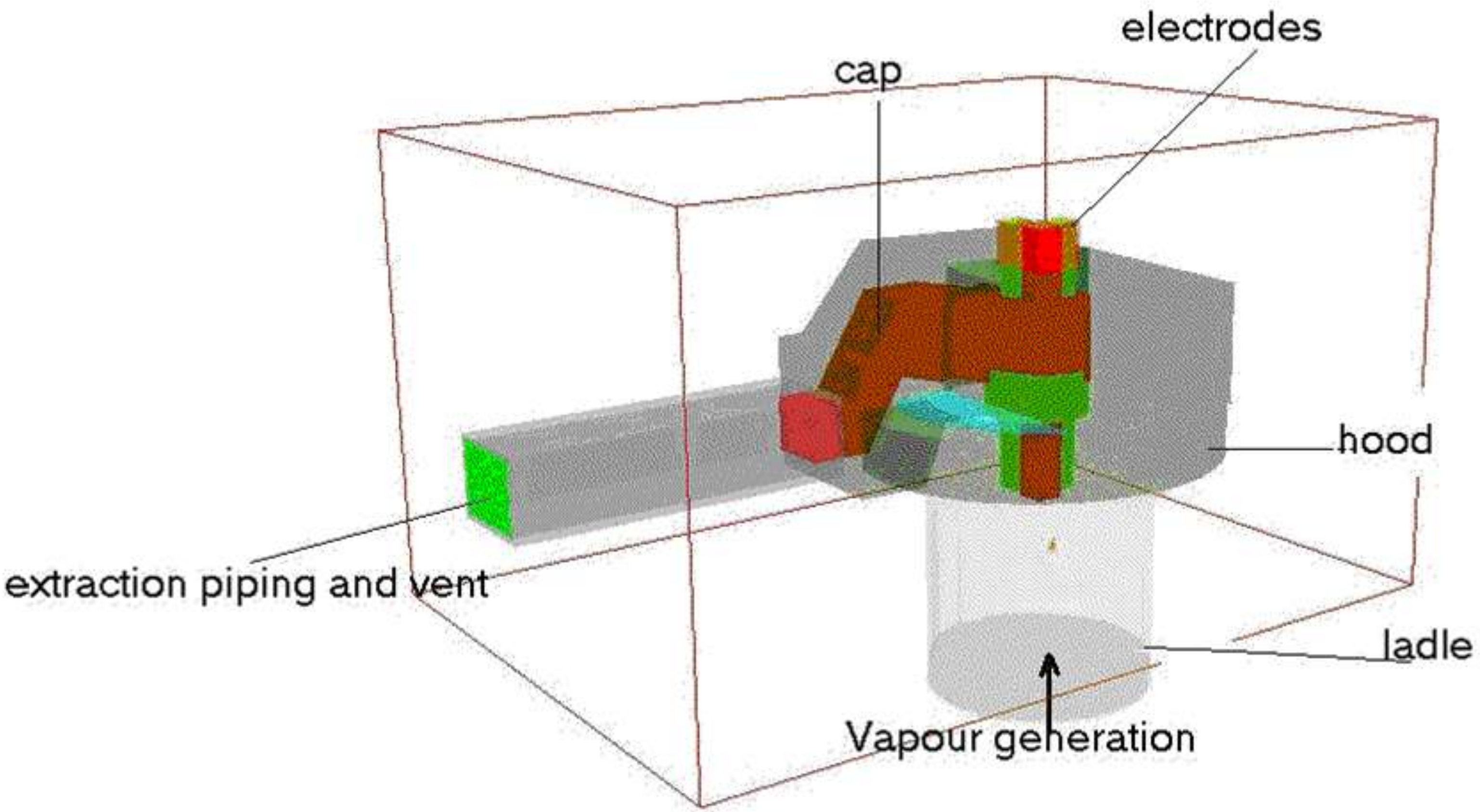
- Metal vapour is generated in a ladle and rises due to buoyancy and forced extraction through a collecting hood and onwards into the cap from where it is sucked along piping at a specified extraction rate.
- Air can be expelled or entrained into the system at various places in the system, namely through gaps between;
  - ladle and hood;
  - electrodes and cap;
  - hood and cap or;
  - exit piping and cap
- The aim of the project is to calculate the proportion of metal vapour that escapes the system under different extraction rates
- The project modelled three different cap designs.

Vapour Extraction - Layout

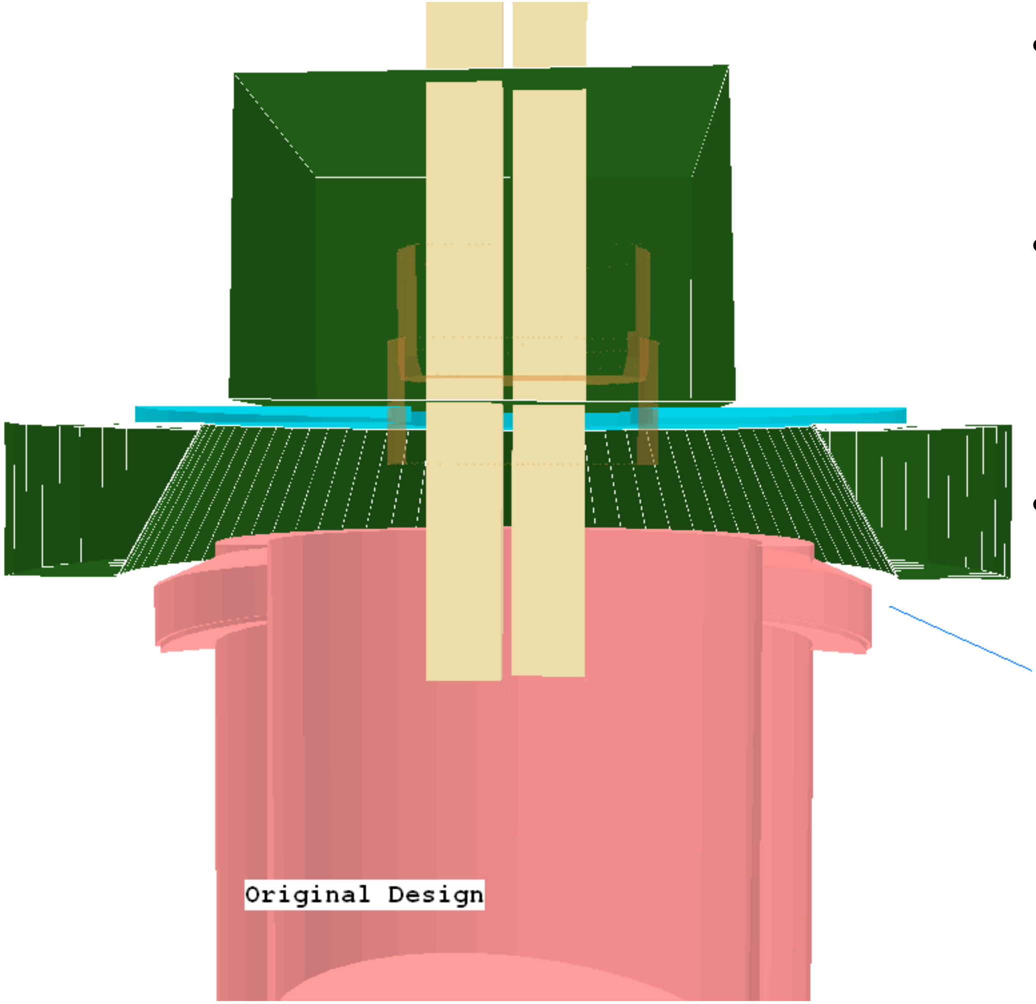
Seminar

CHAM

- Sketch of ladle / hood layout

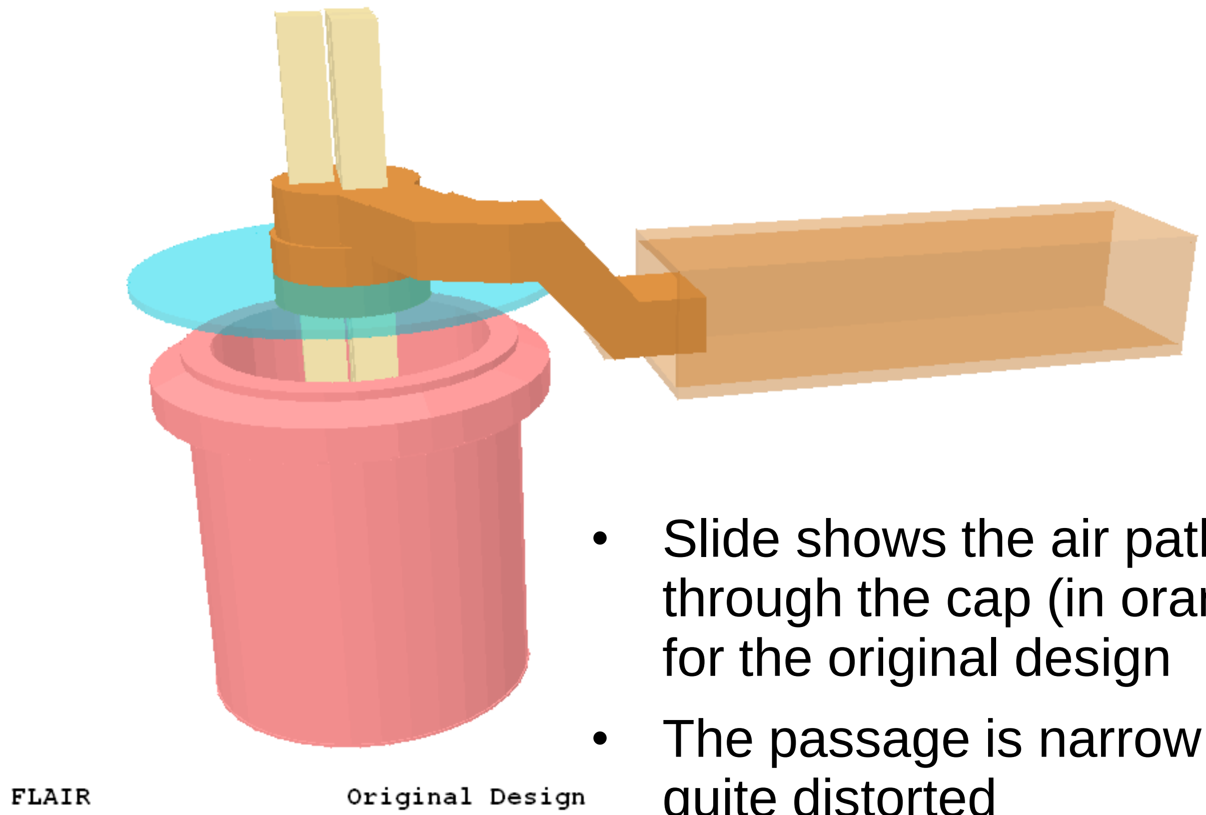


Vapour Extraction – Geometry



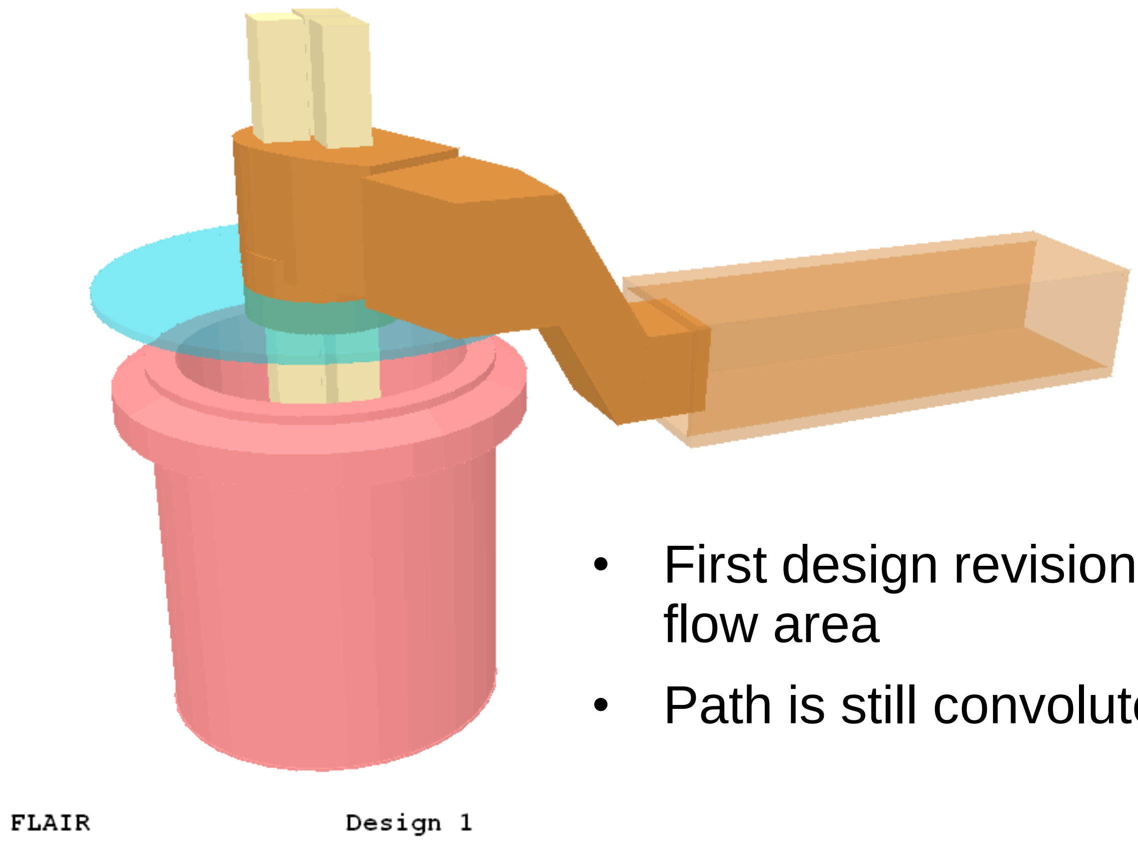
- The slide shows a section through the ladle.
- The lid (blue) and hood (green) are common to all designs
- The hood is omitted from the following slides for clarity.

Vapour Extraction - Original Design



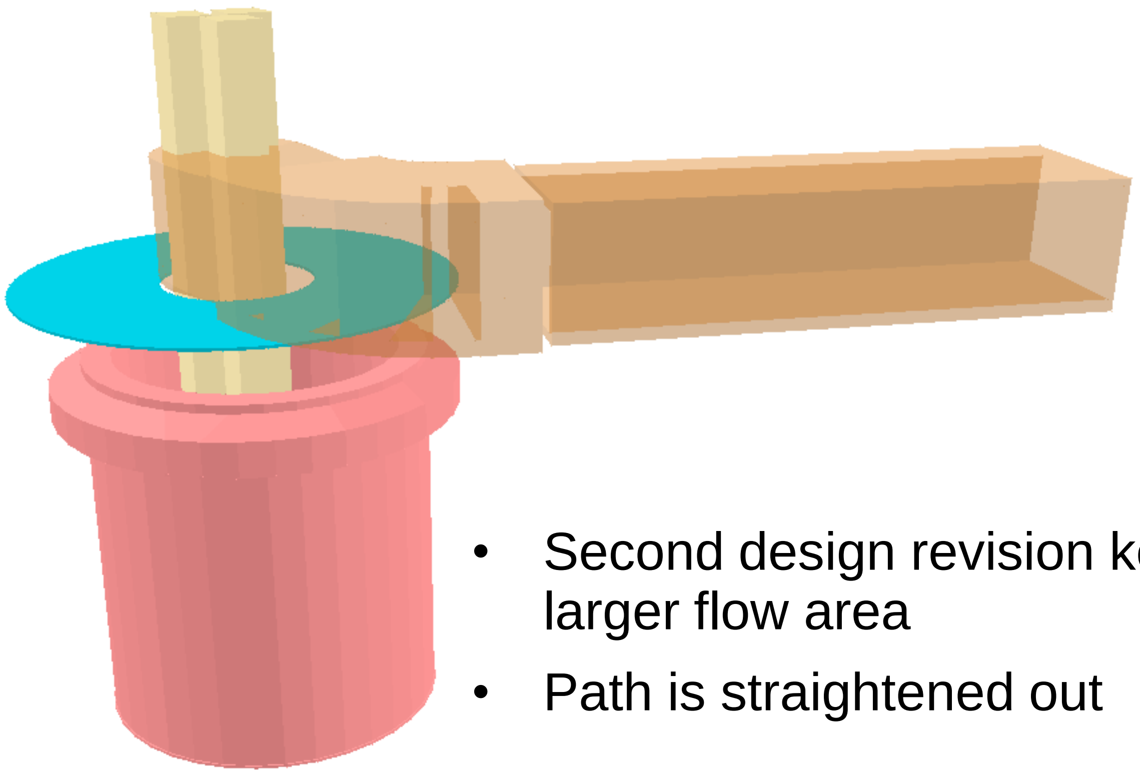
- Slide shows the air path through the cap (in orange) for the original design
- The passage is narrow and quite distorted

Vapour Extraction – Design 1



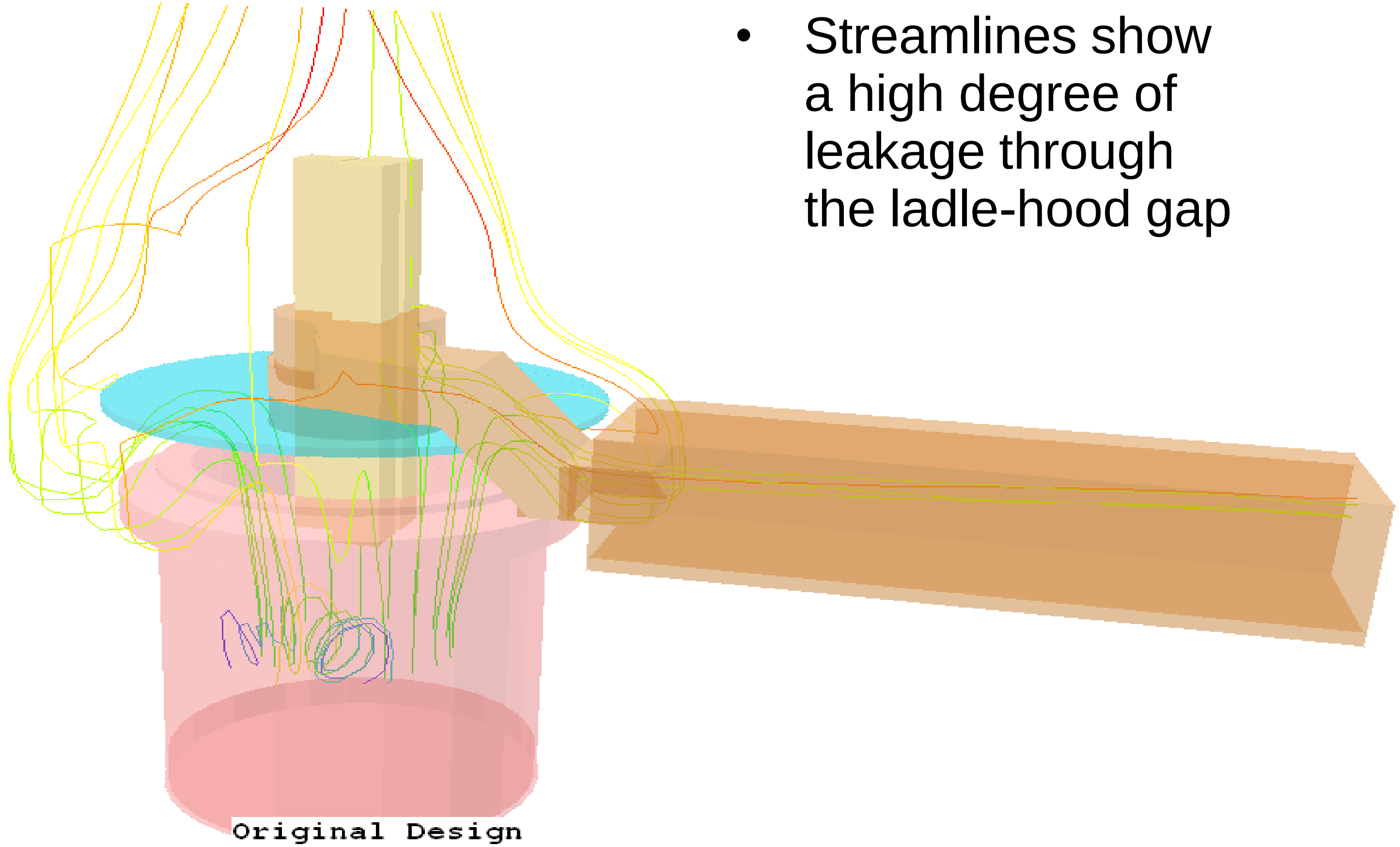
- First design revision has larger flow area
- Path is still convoluted

Vapour Extraction – Design 2



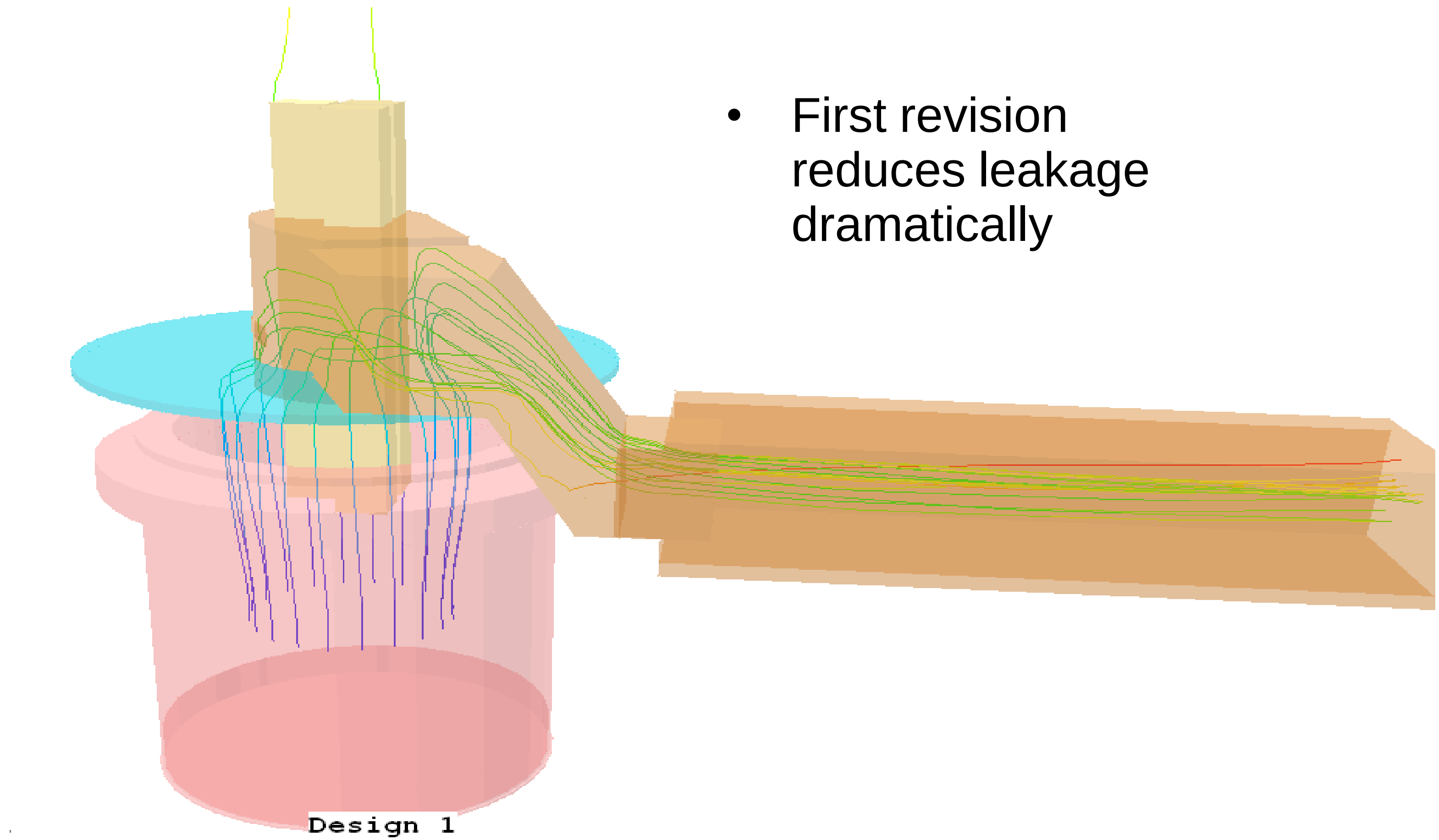
- Second design revision keeps larger flow area
- Path is straightened out

Vapour Extraction - Results



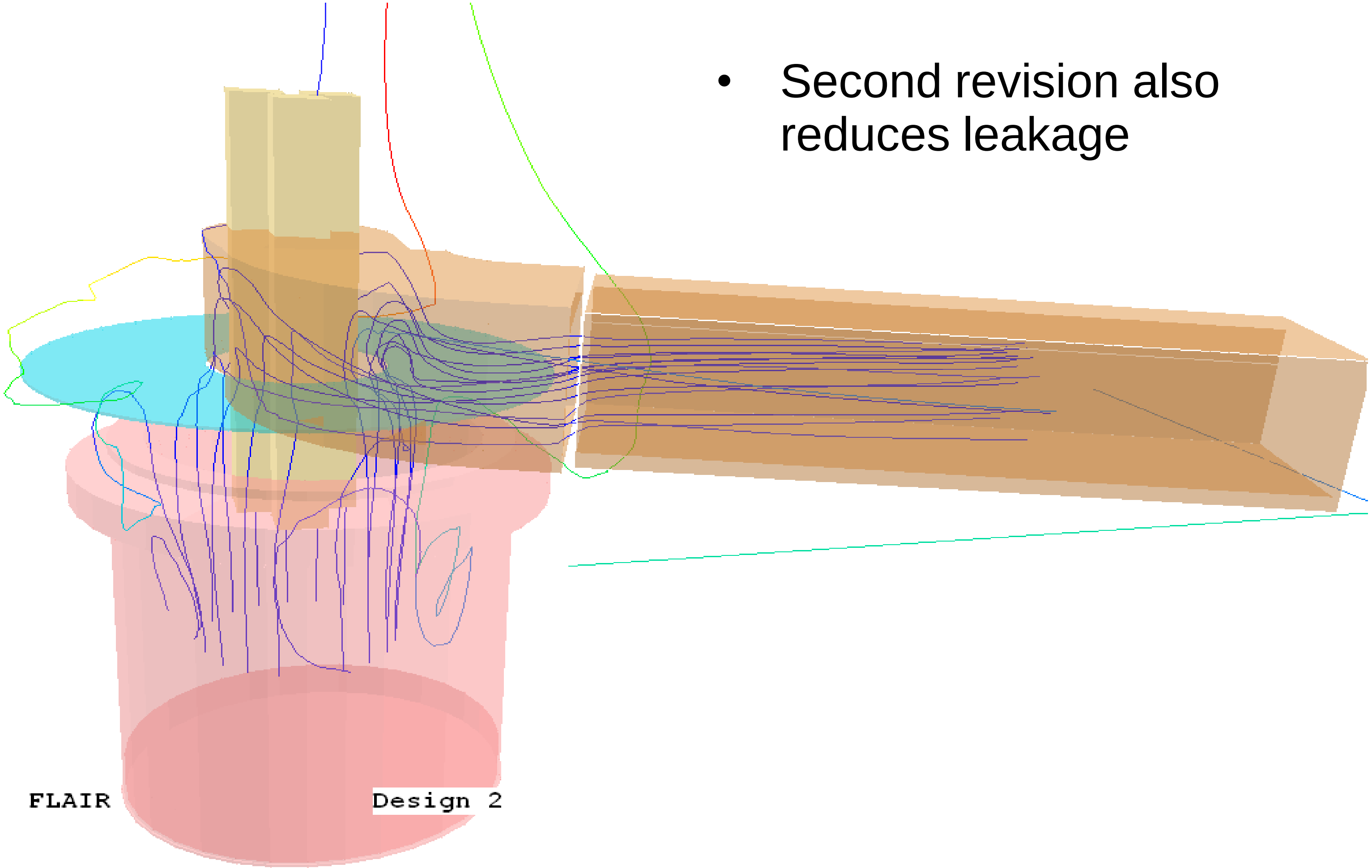
- Streamlines show a high degree of leakage through the ladle-hood gap

Vapour Extraction - Results



- First revision reduces leakage dramatically

Vapour Extraction - Results

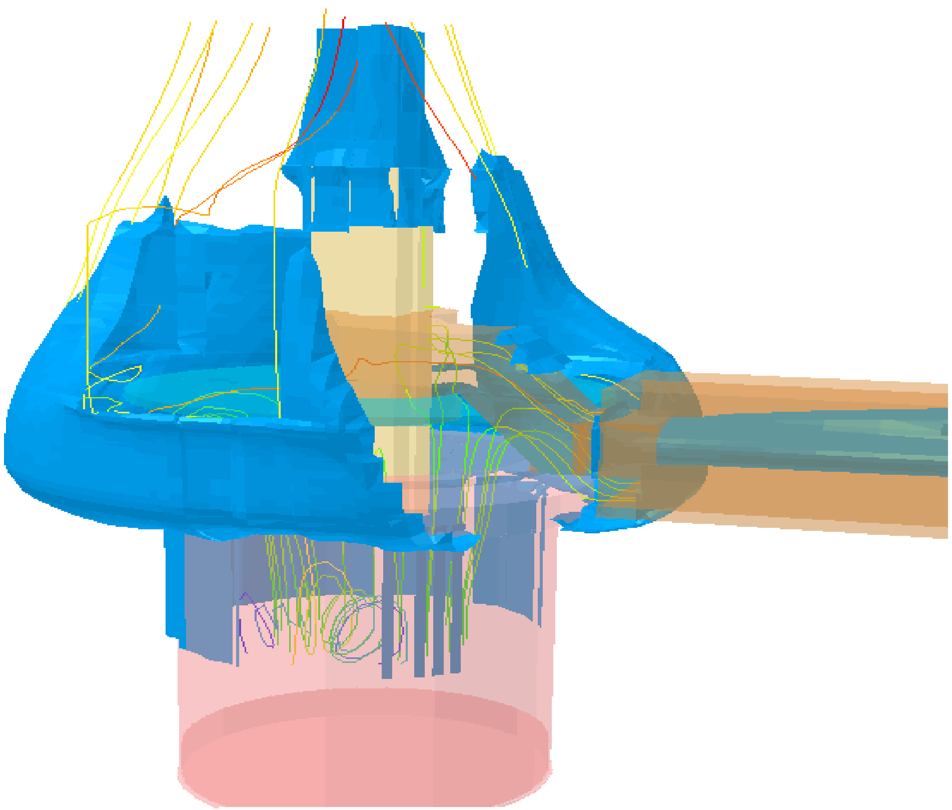


- Second revision also reduces leakage

Vapour Extraction - Results

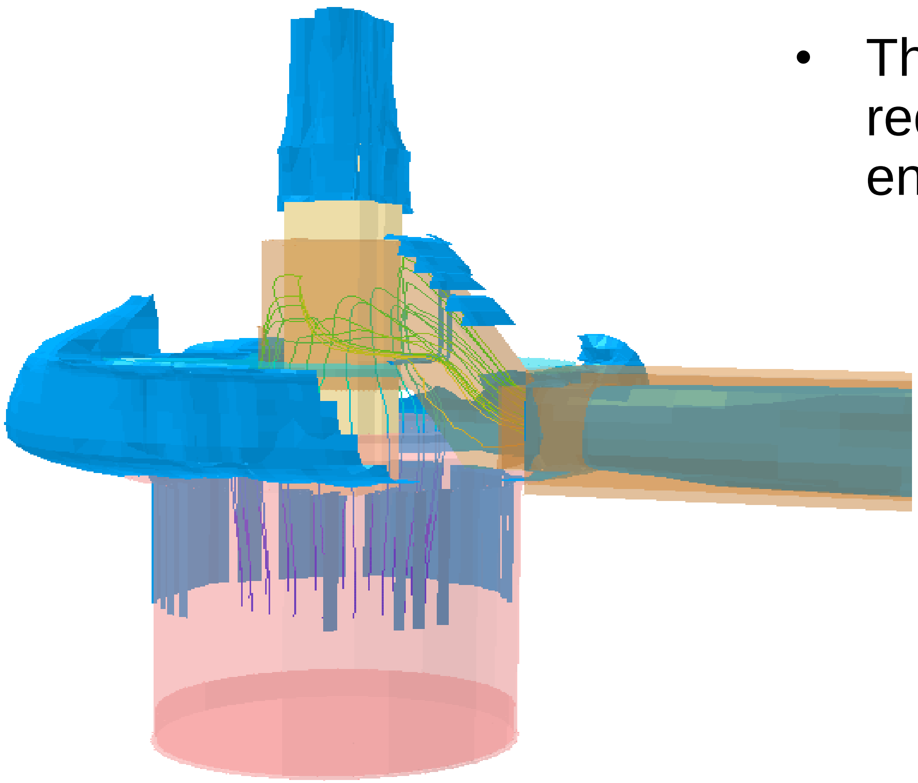
Seminar

CHAM



- An iso-surface of metal vapour of 200,000ppm shows the vapour cloud around the ladle

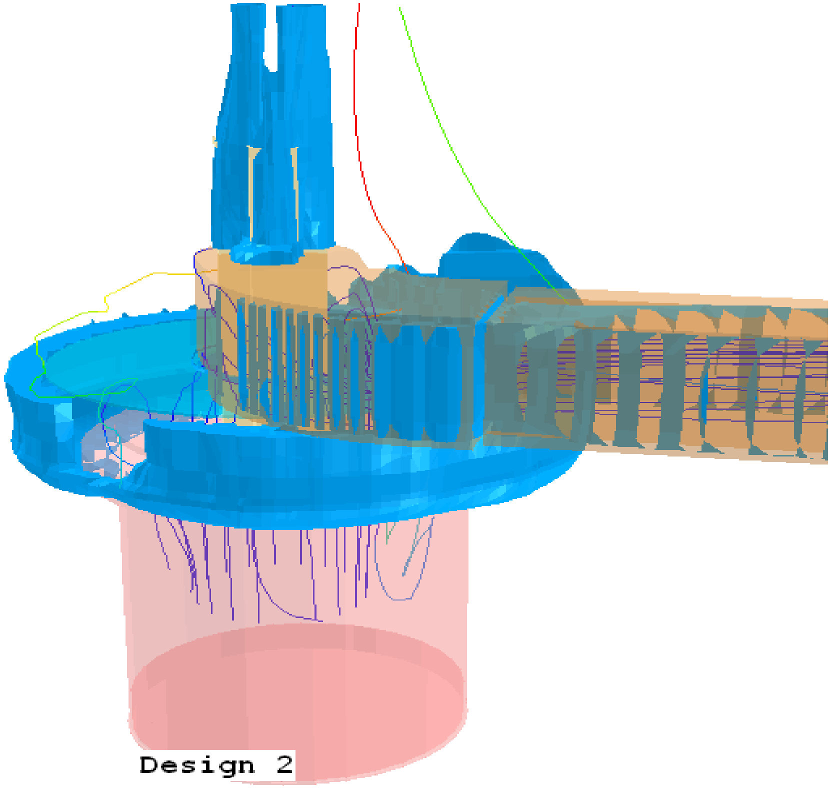
Vapour Extraction - Results



Design 1

- The first design revision reduces the volume of the envelope

Vapour Extraction - Results



- The second revision reduces it even further, keeping it almost entirely confined within the hood

Vapour Extraction - Results

The table below gives vapour proportion to escape the system for each case.

Exhaust rate	65,000 c.ft.m	80,000 c.ft.m	100,000 c.ft.m
Original	59.4%	51.0%	41.6%
Design 1	36.8%	25.0%	10.6%
Design 2	Not available *	Not available *	<1%

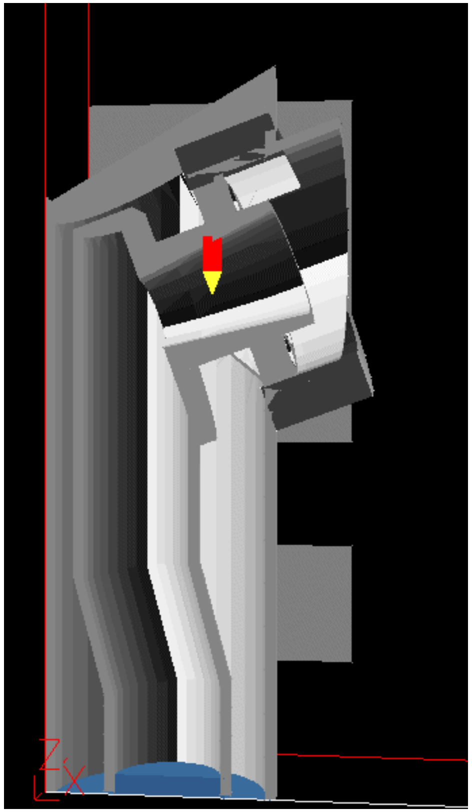
CHAM

PHOENICS
Burner Example

Burner

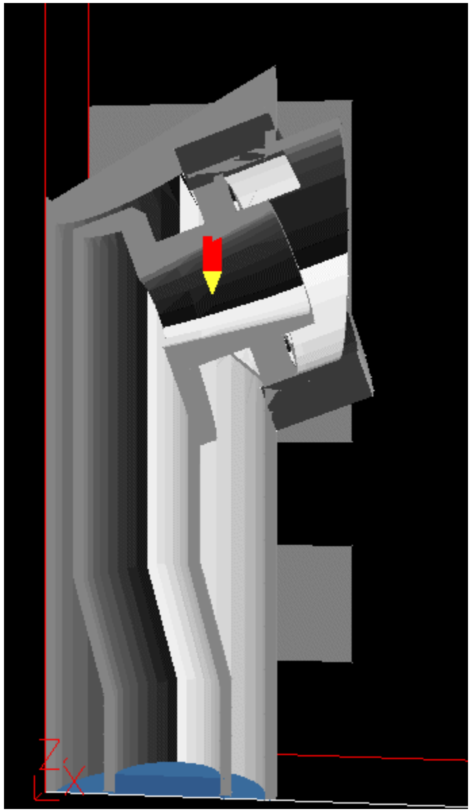
CHAM

- CHAM was asked to make flow calculations for a particular design of furnace burner.
- This burner had proved troublesome in operation, and had already been the subject of development work aimed at improving its performance.
- The geometry was presented to CHAM as an engineering drawing.
- This was turned into an AUTOCAD solid model, and then exported as an STL file.



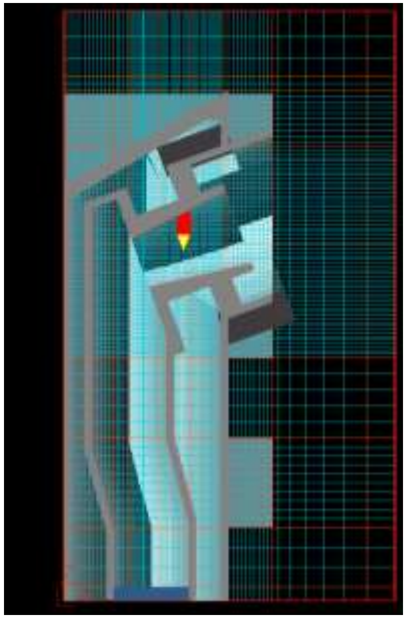
Burner - Geometry

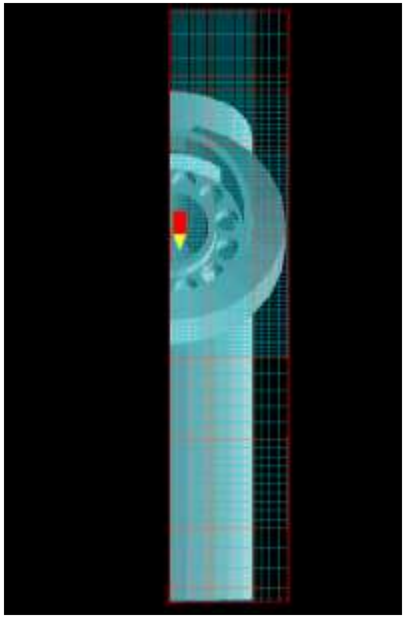
- The burner has a concentric inlet pipe, with fuel fed through the inner pipe, and oxygen in the outer annulus.
- The fuel passes through a nozzle, and the oxygen through a series of holes before finally mixing and combusting.



Burner - Geometry

- In order to resolve the details of the geometry, a relatively fine grid of 80 * 43 * 94 cells was used.





- As can be seen, grid has been concentrated in the region of the nozzle and holes, and also where the inner pipe crosses the mesh at an angle.

Burner - Modelling

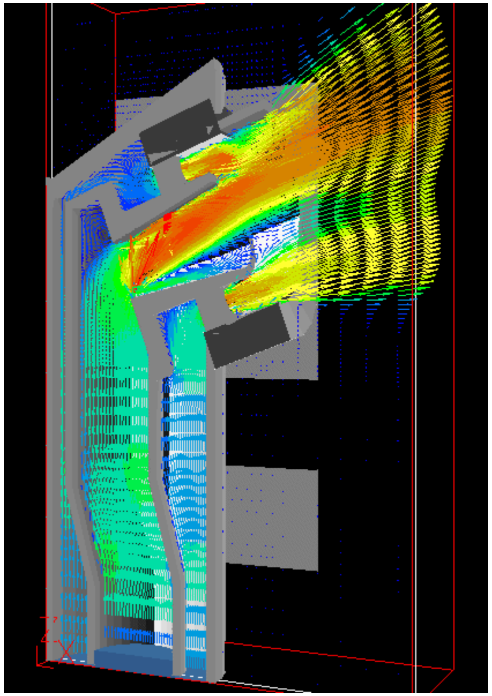
Seminar

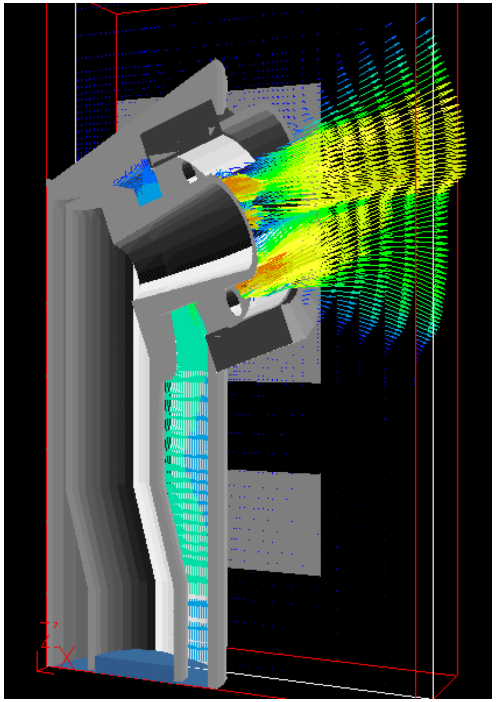
CHAM

- Initially isothermal calculations were made.
- The density was calculated from the Ideal Gas Law, with a mixture-dependent molecular weight to represent the effect of fuel and oxygen mixing.
- Later, calculations were also made using the Simple Chemical Reaction Scheme (SCRS) combustion model, with the reaction rate controlled by Eddy-Breakup
- The standard k-e model was used in all calculations.

Burner – Flow field

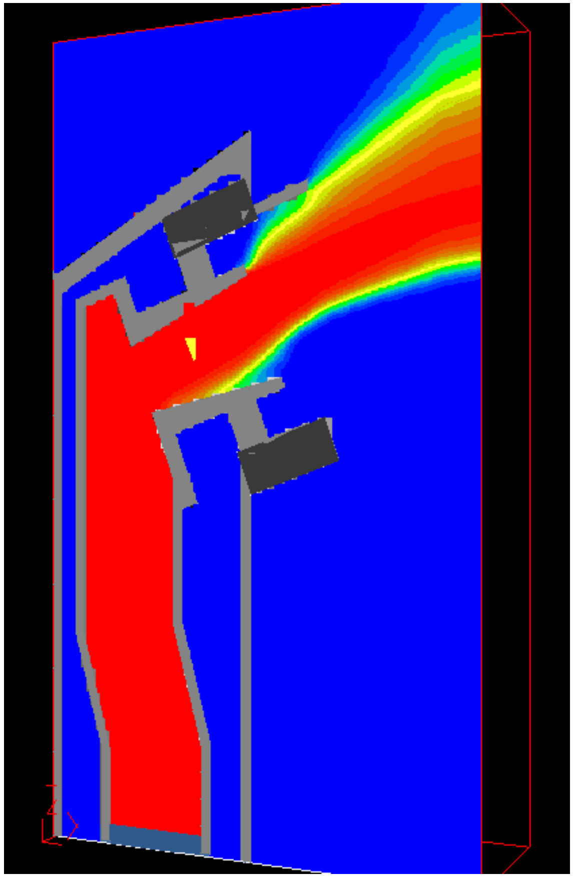
- The flow field shows a strong recirculation zone in the lower part of the fuel nozzle.
- This feature was observed in both isothermal and combusting cases





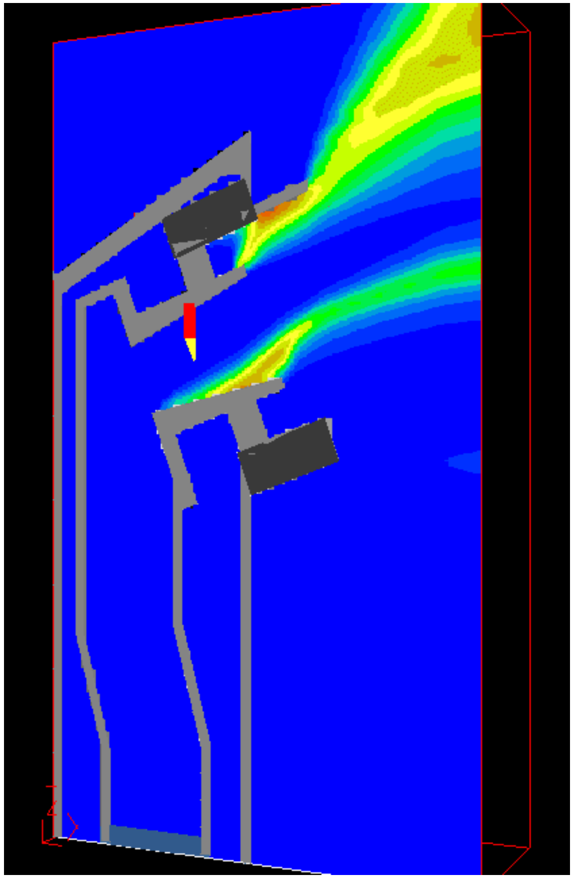
Burner - Results

- This slide shows the fuel concentration on the centre-plane.
- The fuel is depleted along the lower inner nozzle surface



Burner - Results

- This slide shows the temperature at the same location - very high temperatures can be seen inside the nozzle.



Burner - Validation

- The original design of this burner, which is modelled here, also exhibited severe recirculation in the lower part of the fuel nozzle.
- This was confirmed by pressure measurements, which showed negative pressure in this area.
- The predictions also show negative pressures here.
- Finally, when the original design burner was 'lit', it melted the lower part of the nozzle and the oxygen holes at the bottom of the inlet ring.
- This confirms that the model is reproducing the effects seen experimentally.

PHOENICS

Mechanical Pump Ring Seal Example

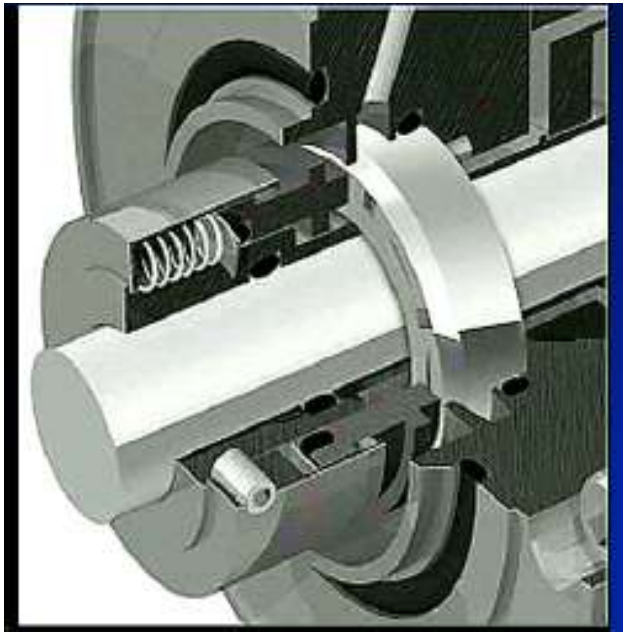
Polar Coordinate Examples
Mechanical Ring Seal

Seminar

CHAM

POLAR PARSOL & Wall Rotation features of PHOENICS applied to a Mechanical Pump Ring Seal problem

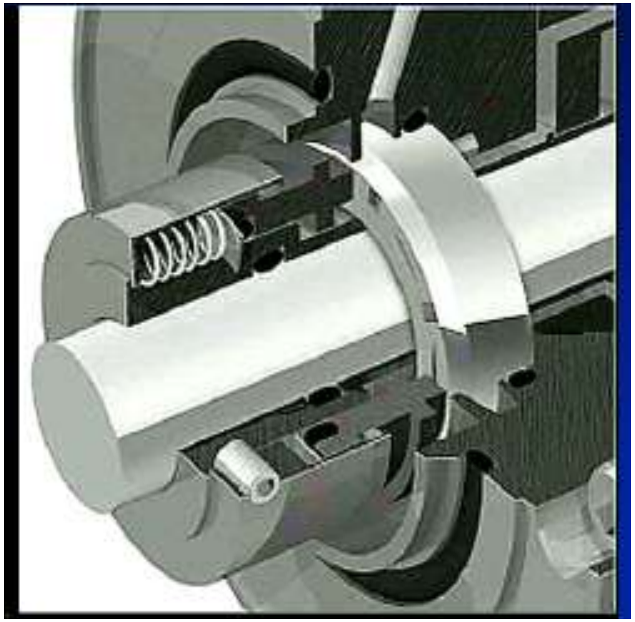
Pump Rings are cooling devices widely used in the sealing systems of rotary machines. They use water circulation to extract heat caused by seal friction. A major concern of engineers is how much heat the cooling system can take, which mainly depends on the flow rate.



Polar Coordinate Examples
Mechanical Ring Seal

The pumped process fluid (liquid or fluid with vapour bubbles or solids) is sealed at the radial face gap between the rotating primary seal ring and stationary mounted mating ring. Frictional heat occurs at the sealing interface.

In this example, the rotation of the device is treated as a “slip wall” on the rotating parts with angular velocity.



Polar Coordinate Examples

Mechanical Ring Seal

CHAM

The Pump Ring with a seal system on the axis is shown in Figure 1. The yellow cylinder above the device is an inlet and the blue one below the device is the outlet. The rotating Pump Ring and mating ring are shown in Figure 2.

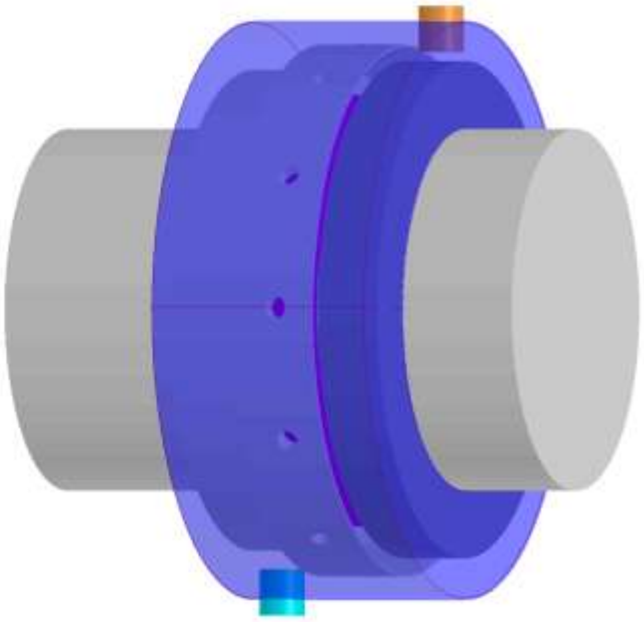


Fig 1. Pump Ring with seal system

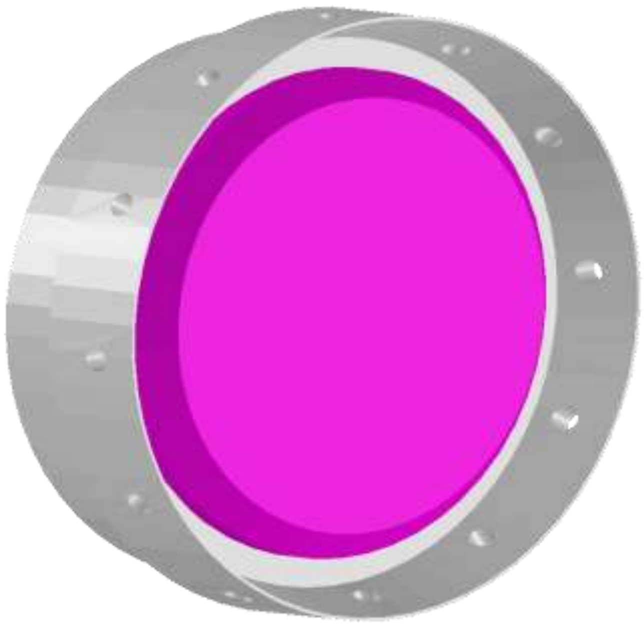


Fig 2. Geometry of Pump & Mating Rings

Polar Coordinate Examples

Mechanical Ring Seal

Seminar

CHAM

Seven simulations were carried out which produced the pressure and flow rate curve shown below.

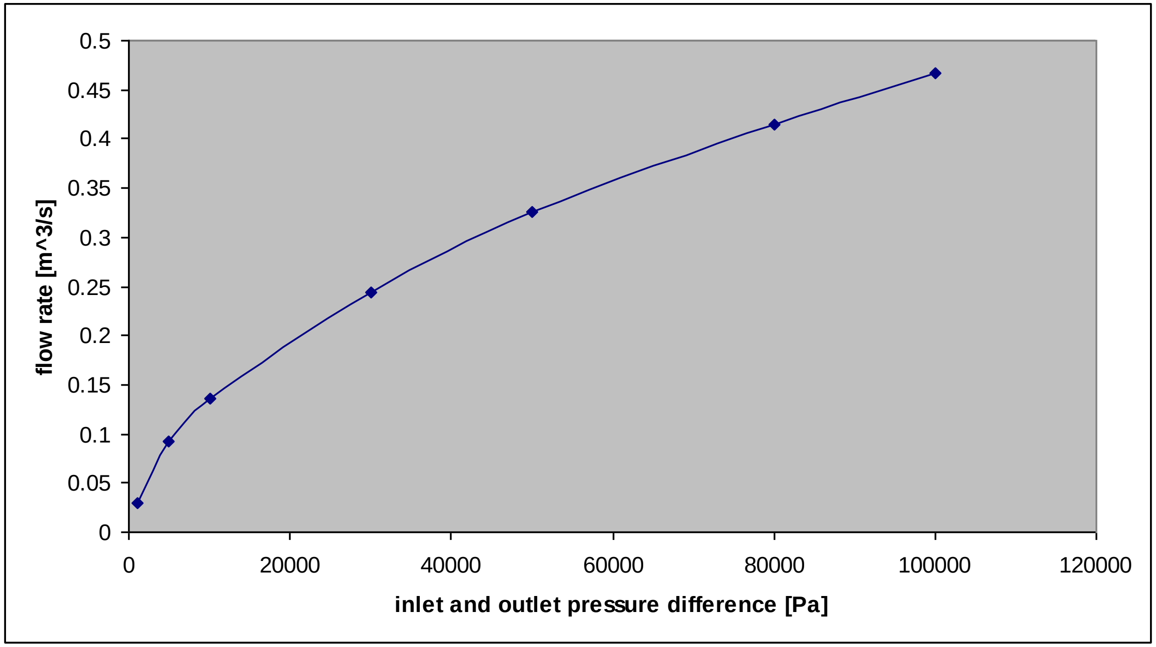


Fig 3. The relation of Pump Ring flow rate with pressure

Polar Coordinate Examples

Mechanical Ring Seal

CHAM

The velocity vectors below clearly show the coolant coming from the inlet along the stationary surface of the primary ring into the passage between the Mating Ring and the Pump Ring, and then flowing through the hole on the Pump Ring to the outlet.

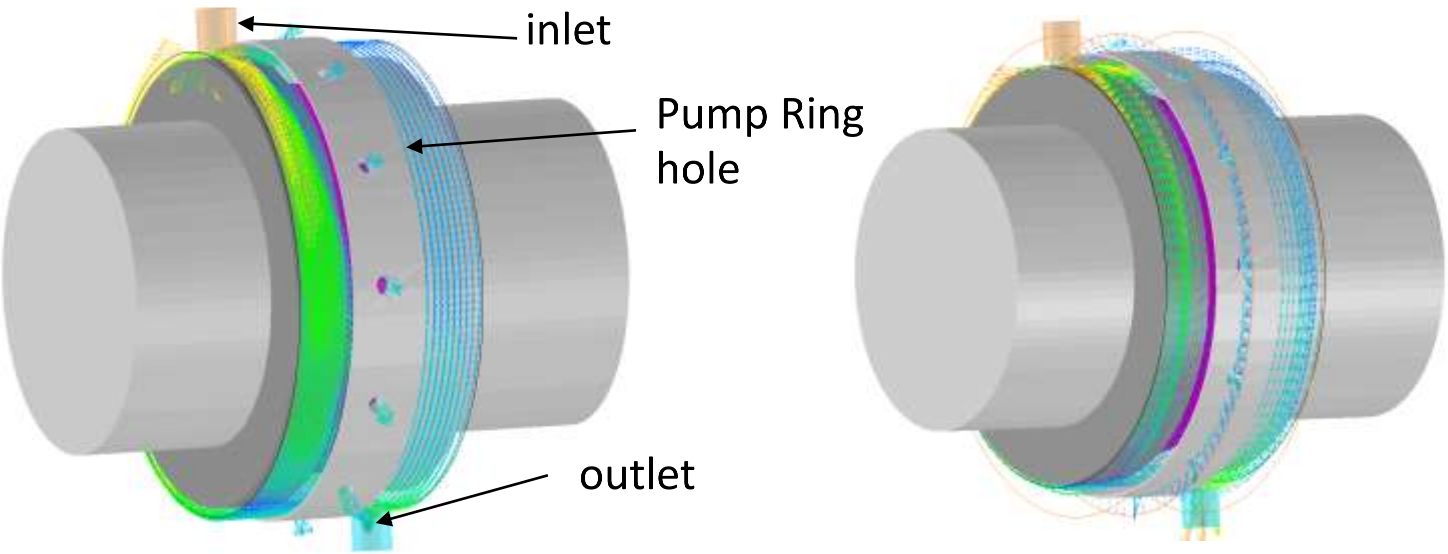


Figure 4. Velocity vectors on Y-plane near inner surface of Pump Ring (left) and X-plane along the Pump Ring hole (right)

Polar Coordinate Examples
Mechanical Ring Seal

Seminar

CHAM

Pressure contours on the Pump Ring surface in Fig 5 indicate high- pressure areas on the inside surface of the Pump Ring. Fig 6 shows streamlines of the flow path coloured by residence time.

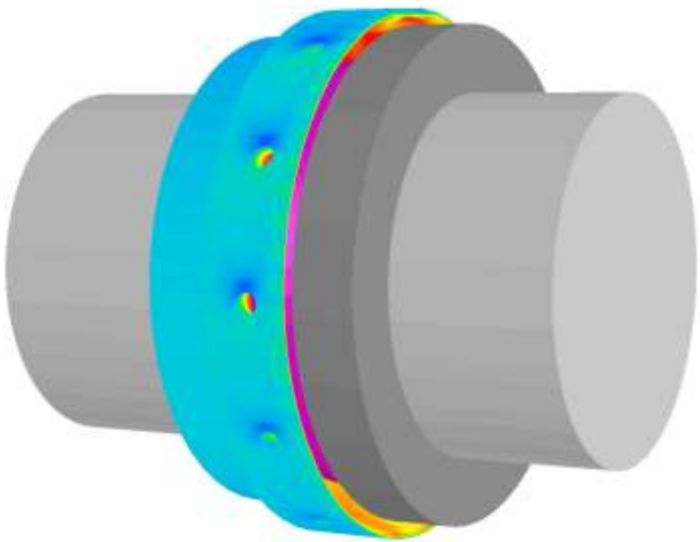


Fig 5. Surface pressure of Pump Ring

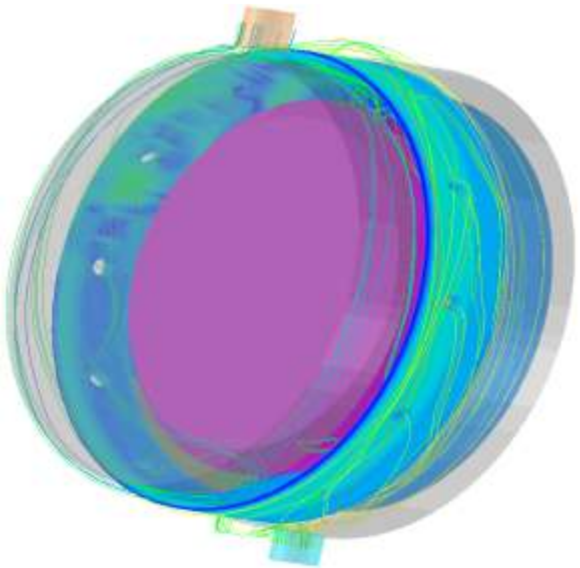


Fig 6. Streamlines of residence time

Polar Coordinate Examples

Mechanical Ring Seal

The case demonstrates how the wall rotation feature in POLAR PARSOL simulates rotary machines.

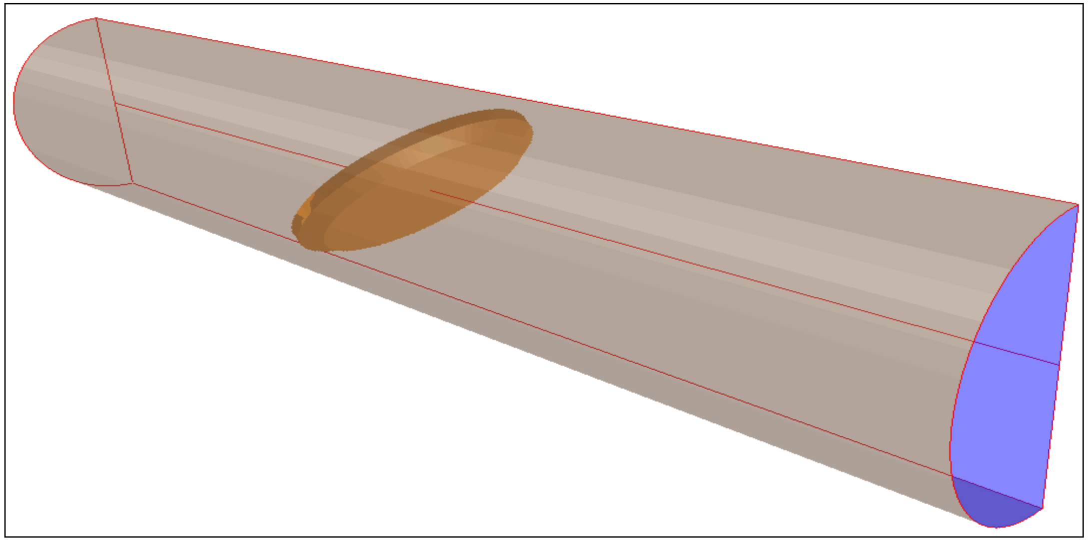
For most steady rotation cases, wall rotation can be over ten times faster than running transient MOFOR.

Additionally, the automatic POLAR PARSOL grid generation makes the analysis work as easy as clicking a button.

Polar coordinate cases
Butterfly Valve

CHAM

A Butterfly Valve in a pipe is a typical POLAR case for CFD applications. The flow rate through the pipe is controlled by the rotation angle of the valve.



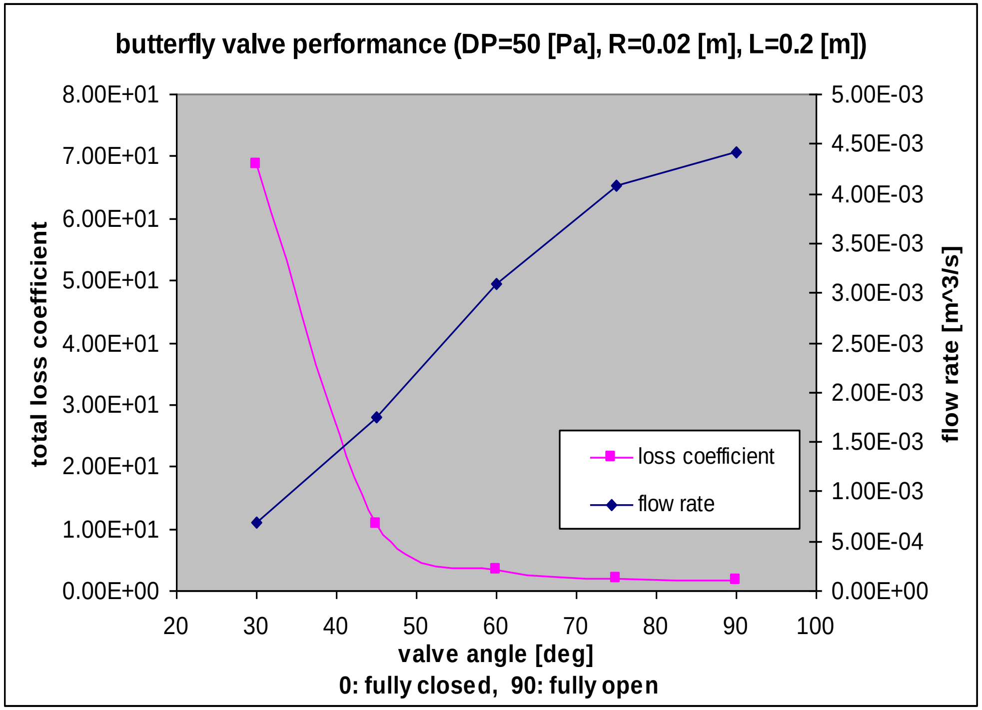
Control engineers need to know how sensitive the flow rate is to fluid pressure and valve angle changes and the consequential implication for hydraulic loss.

Polar coordinate cases
Butterfly Valve

CHAM

The demonstration case uses a 40mm diameter of 4mm thickness. The pipe length is 200mm.

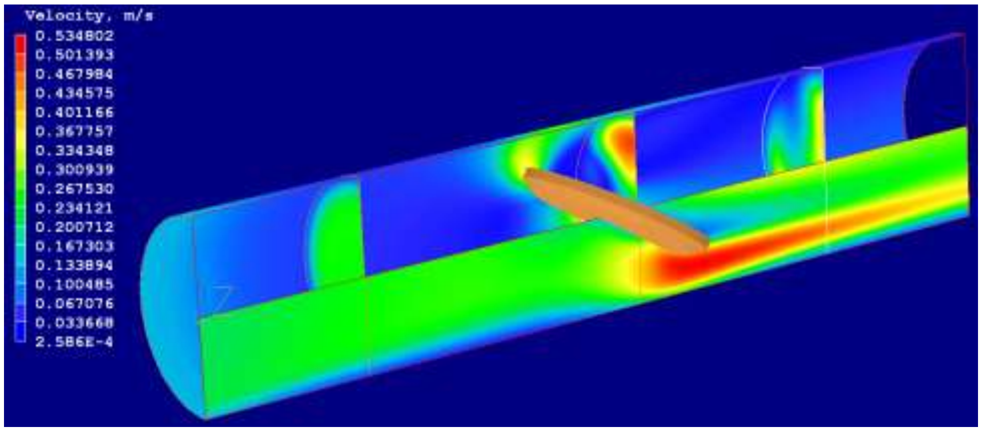
Five runs were carried out with different valve openings (with angle 0^o fully closed and 90^o fully open). The relative working pressure used is 50 [Pa] at the inlet and zero at the outlet.

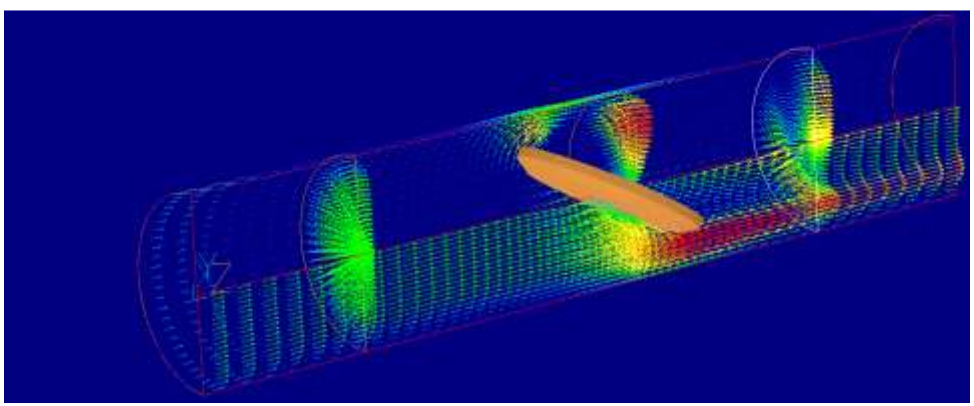


The results shown give the relationship between valve angle, total loss coefficient and flow rate. The total loss coefficient is high with a small valve opening, and decreases by nearly 90% during the first 45^o opening. Correspondingly, the flow rate is almost linearly increased below 50^o, before slowly increasing at even wider apertures.

Polar coordinate cases
Butterfly Valve

CHAM



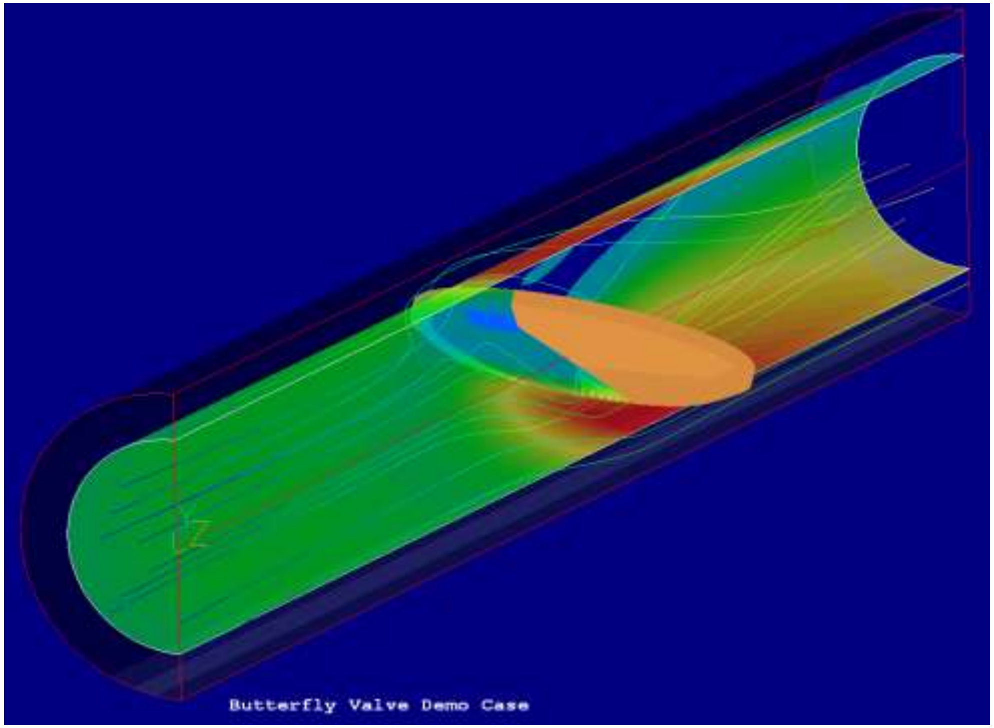


Velocity contours with a 45^{o} opening in Figure 3 show the incoming stream deflected by the valve.

The highest velocity 0.53 [m/s] at the bottom opening results in jet flow and fluids through the top gap spreading along the whole pipe section, as expected.

Streamlines of residence time in Figure 4 show the same flow pattern.

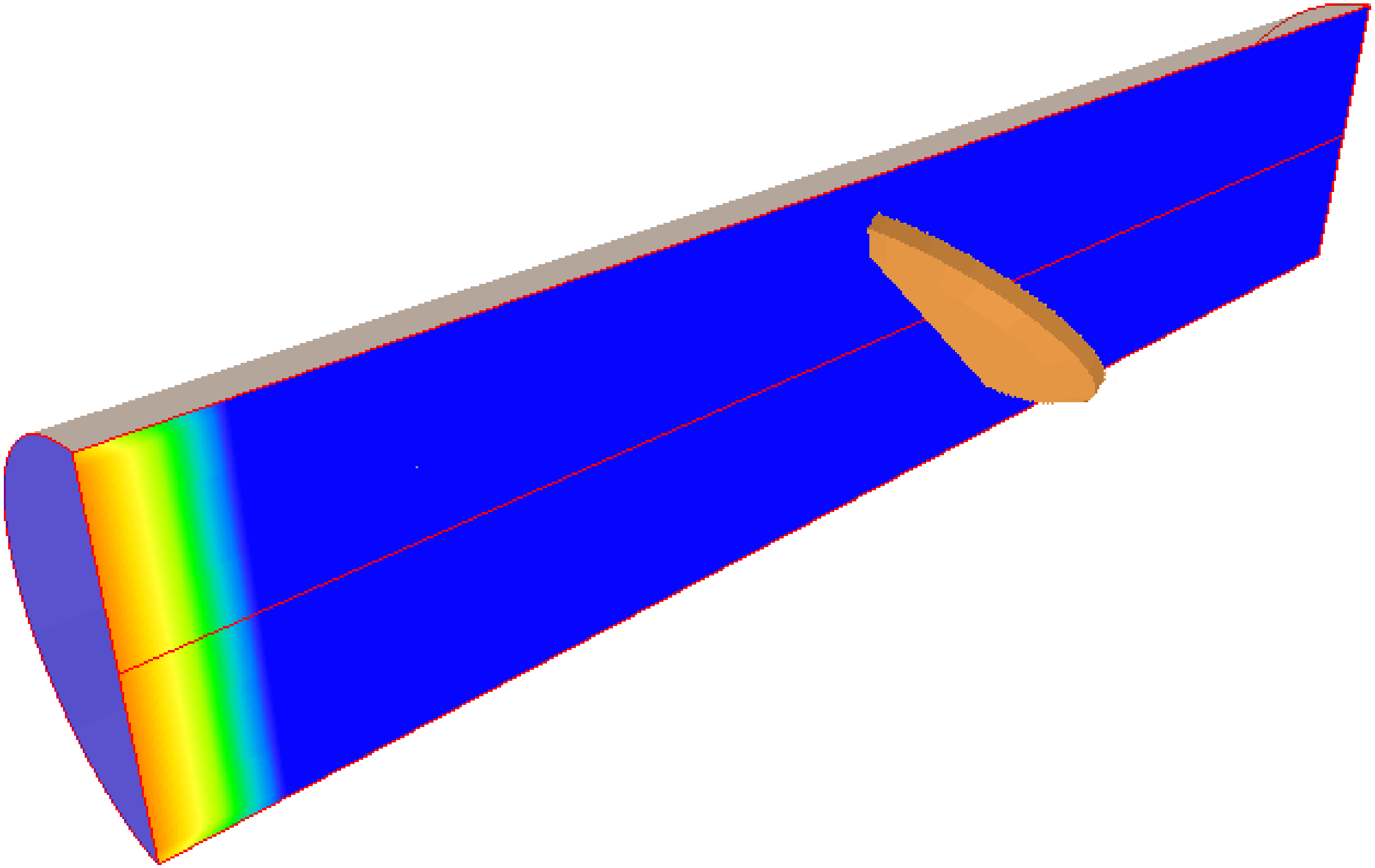
Polar coordinate cases Butterfly Valve



Streamlines of residence time at 45° valve opening

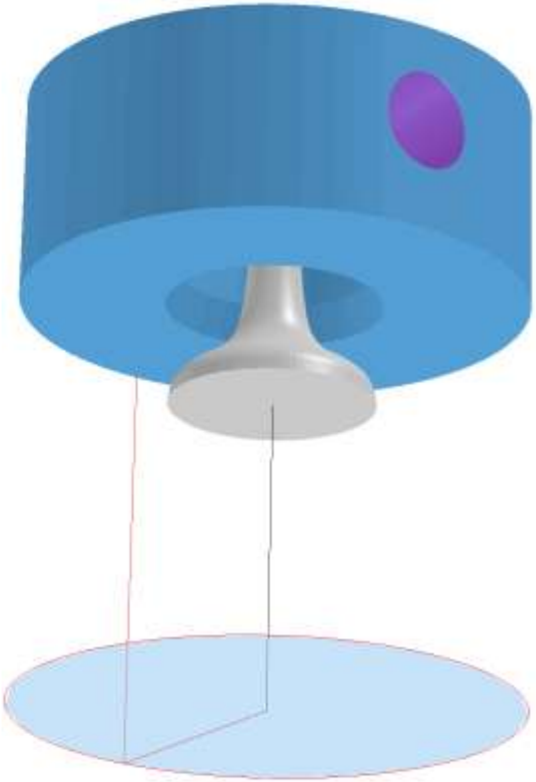
The simulation demonstrates that the POLAR PARSOL feature is a useful tool for valve designers to predict the valve performance to achieve optimal design, and allows control engineers to analyse and improve the behaviour of valves under differing operational circumstances.

Polar coordinate cases
Butterfly Valve



Polar Coordinate cases
Poppet Valve

The poppet valve motion in an internal combustion engine is of particular importance to design engineers, and has been widely studied. This example demonstrates the ease-of-use of the “POLAR PARSOL” feature in PHOENICS when combined with “MOFOR”



The poppet valve geometry shown in grey moves up and down periodically in the centre of a seat ring, shown in blue. The inlet is on the surface of the seat ring, and the entire base is an outlet. In this simulation, the diameters of the valve head are 40mm, the valve stem: 12mm, the chamber: 50mm, and height 80mm.

Polar Coordinate cases
Poppet Valve

The simulation was run for 0.04 [s] in 10 time steps. The motion of the valve in the axial direction is linear, within a distance of 22 [mm]. Results at the sixth time step are provided.

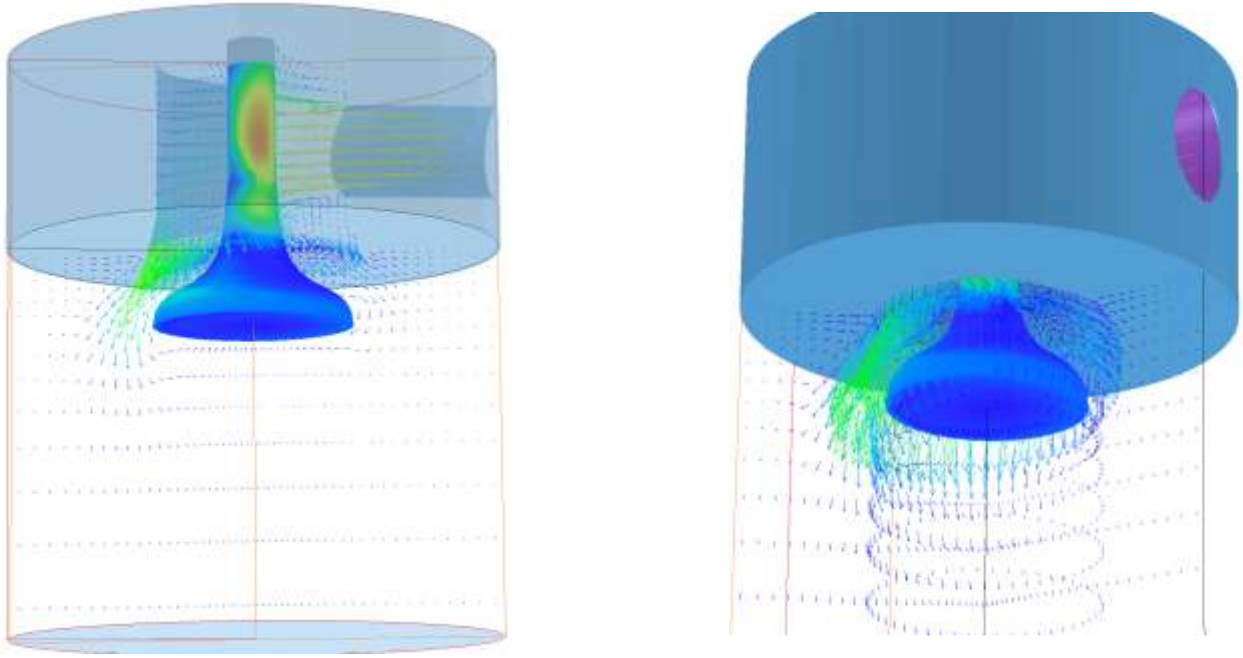
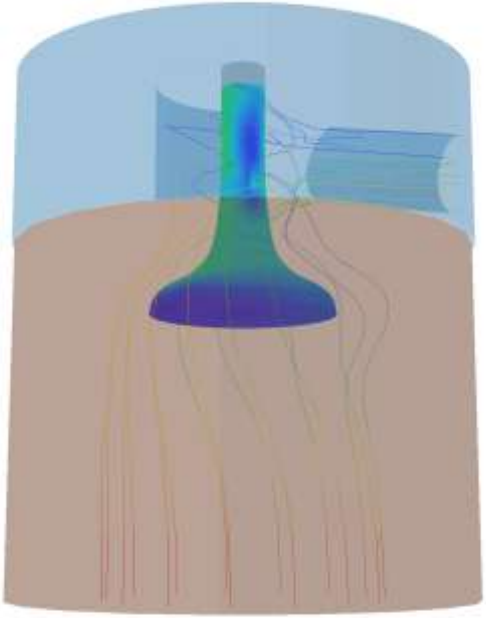


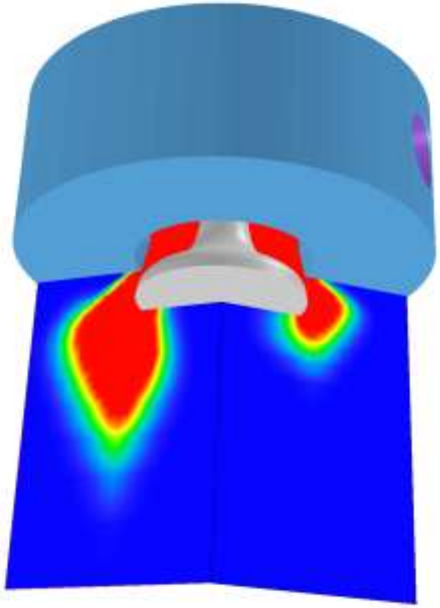
Figure 6 shows the fluid coming from the side inlet on the seat ring then being pushed out through the gap between valve head and seat ring.
Weak circulation occurs in the chamber just below the seat ring.

Polar Coordinate cases
Poppet Valve

CHAM

The main flow stream moves straight towards the outlet, as shown below. The scalar tracer in the next image shows the same flow pattern.



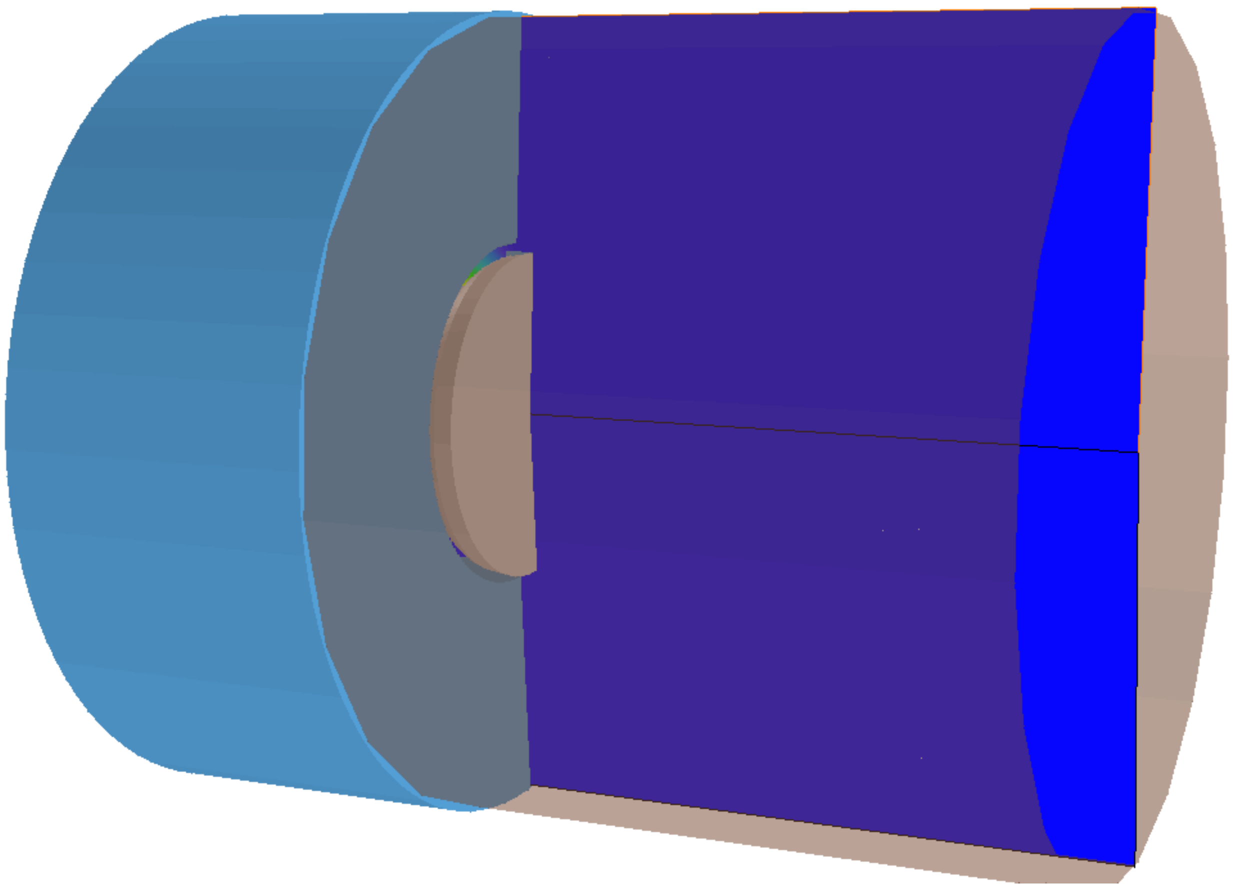


The example demonstrates that the combination of POLAR PARSOL and MOFOR can help the design engineer analyse fluid flow scenarios to optimise valve design or control parameters.

Polar Coordinate cases
Poppet Valve

Seminar

CHAM



Solidification of Silicon in a Crucible Furnace

Demonstration example of PHOENICS simulating the unsteady growth of a silicon ingot within a crucible furnace. The problem considered involves the simultaneous analysis of the thermal solid stress distributions in the Si-solid.

As the model involves the solidification of silicon over time, a transient analysis is necessary. Experience has shown that the actual operating time to complete solidification is about 30 hours.

The geometry is defined using cylindrical polar coordinates using dimensions supplied by the client; the initial temperature condition is set at 1500˚C, and the special properties of silicon introduced via the PROPS file.

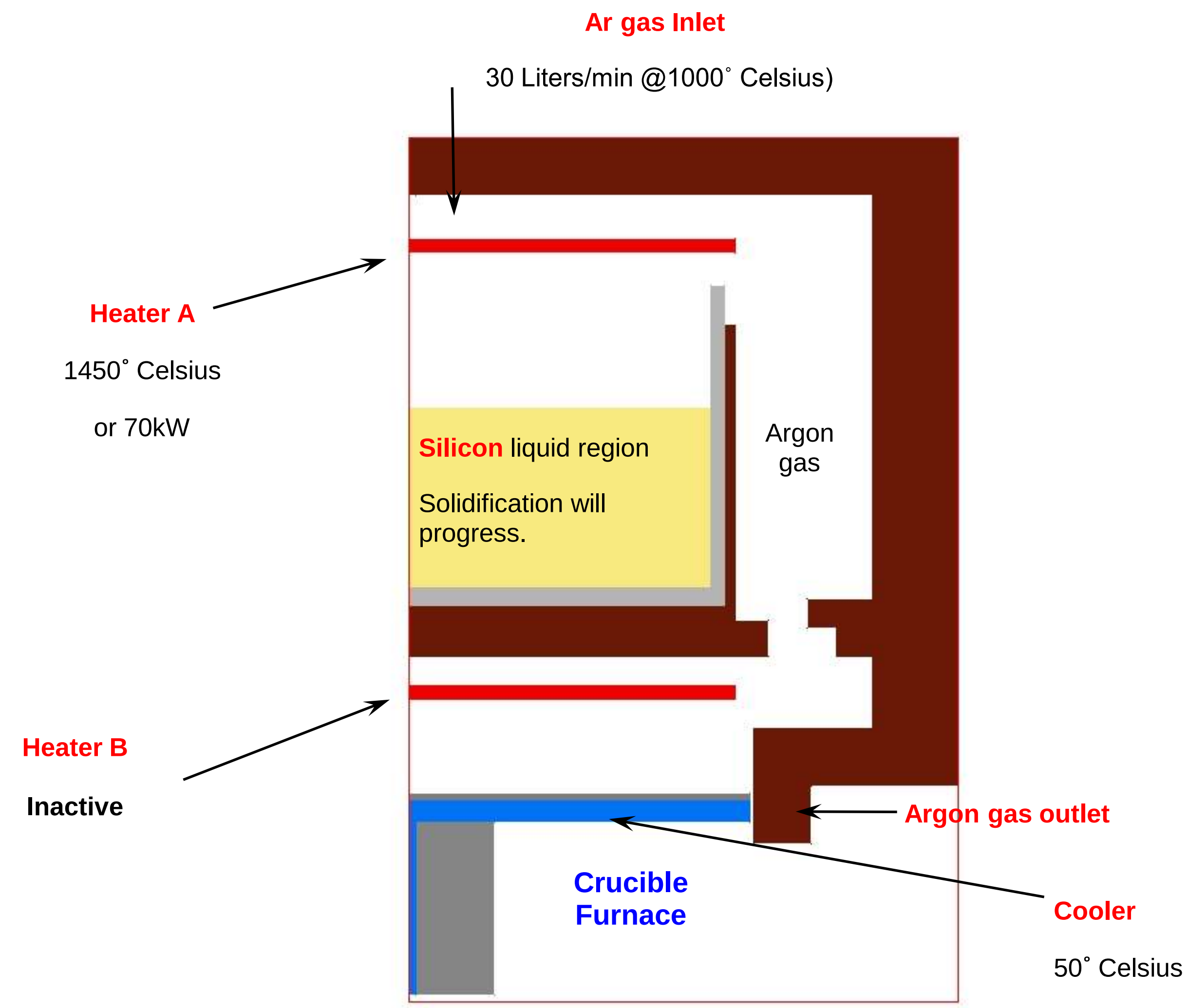
Properties of Silicon

	Liquid	Solid
Density [kg/m3]	2550	2490
Cp [J/kg▪K]	1222	670
Conductivity [W/m▪K]	60.6	84
Viscosity [N▪s/m2]	7.56E-4	
Melting point	1414˚C	
Latent heat	1.809E6 J/kg	

Solidification of Silicon in a Crucible Furnace

Seminar

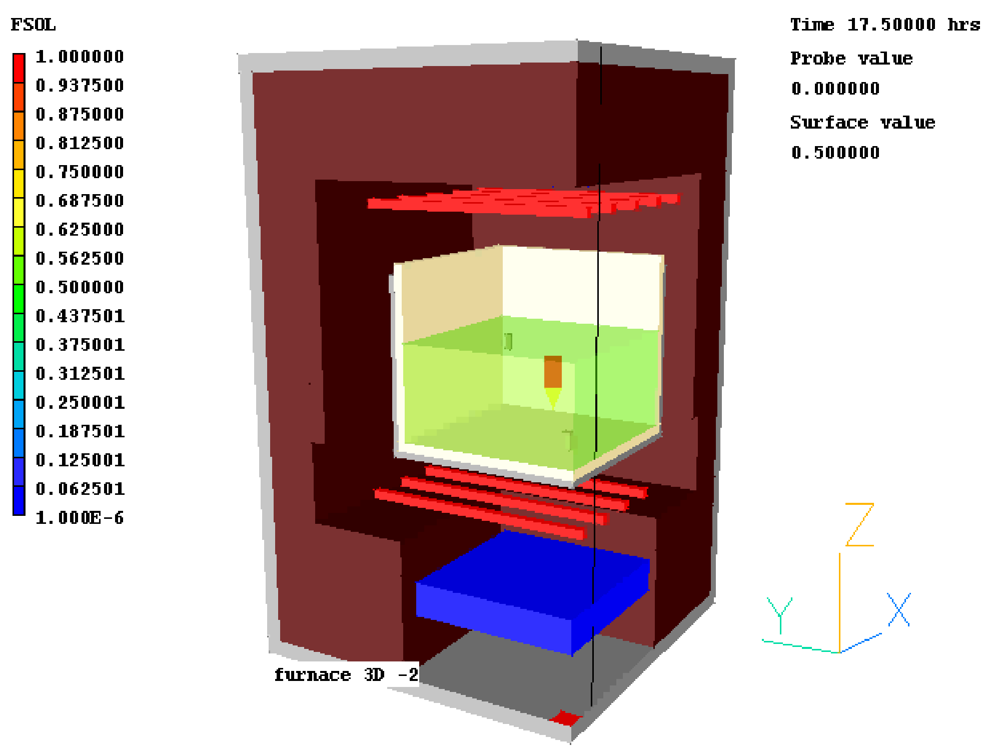
CHAM



Solidification of Silicon in a Crucible Furnace

Seminar

CHAM



Solidification front between 17.5 – 34 hrs

Example Case Studies

The following Application Examples will be shown:

- Office Building Ventilation
- External flow over buildings
- Data Centre Cooling Control
- Urban city planning
- Internal air conditioning systems
- Vent stacks in tunnels

Introduction

PHOENICS/FLAIR was used for the analysis of a multi-storey building in the Kista region of Stockholm, Sweden.

A model was created for testing the internal temperature distribution when subjected to worst-case winter and summer conditions(i.e. very cold or very hot).

In particular, the production of cold downdrafts in the atrium or along the large glassed façades during the winter. There was concern about whether there were regions of unacceptably high air temperature during the summer time.

Introduction

The building design was supplied in the form of a number of AutoCAD.DWG (Drawing) files of the building and its location, along with the operational boundary data, such as:

- the glass specification,
- the building material,
- internal heat sources, together with an estimate of the number of people, and supplementary heating and cooling baffles.

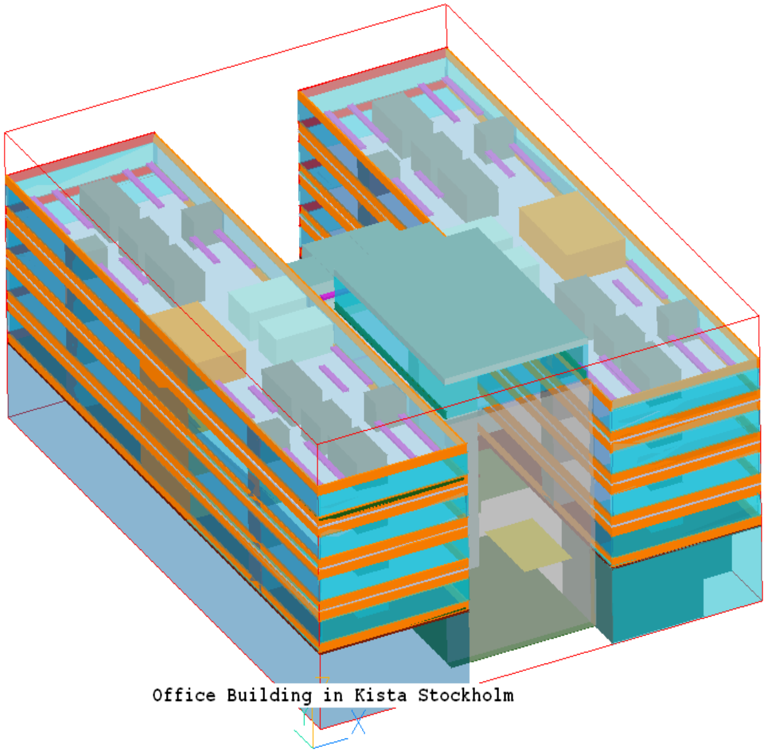
- ➢ AC3D was used to create 'bespoke' objects for the office floor
- ➢ offices are located on four floors on either side of the atrium.

Geometry Creation

Included within the model are some 650 objects representing doors, walls, roof, ceilings, glass windows, computers, persons, office furniture and various types of heat-sources.

The distribution of these objects in all offices on each floor is similar.

Once one floor had been created, the ‘Array Copy’ feature was used to quickly generate the remaining floors.



Problem Specification

For summertime conditions, a solar heat gain of 46,580 Watts is specified through the glass doors and windows, with the radiation projected onto the floors and internal walls.

In addition, there is heat generated by people, lighting and machinery inside the building.

The temperature within the building is regulated by an air conditioning system introducing cooled air at 15°C, and a ventilation system generating a total air exchange of 2300 l/s throughout the building.

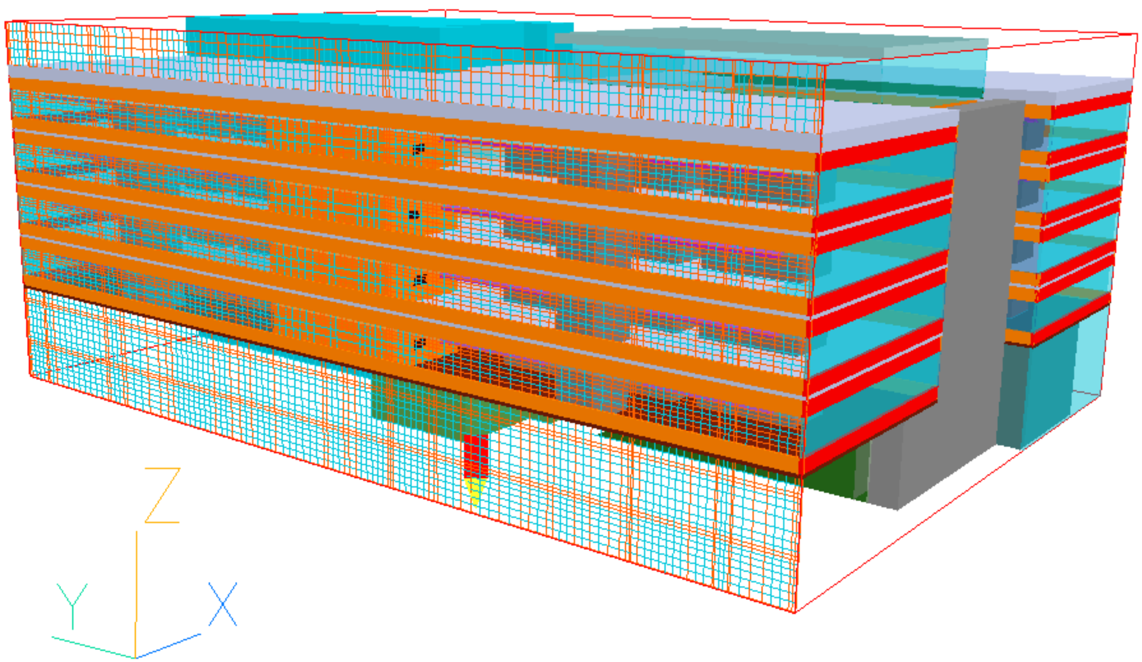
The winter case differs in that there is no solar heat affecting the temperature in the building.

Due to the low temperature outside, the glass door and all the glass windows take heat away from the building.

The temperature of the ventilation air in the building is increased from 15°C to 18°C.

Results

A total mesh size of 1.1M cells (108 * 123 * 85) was used, non-uniformly distributed over the entire calculation domain.



Office Building in Kista Stockholm

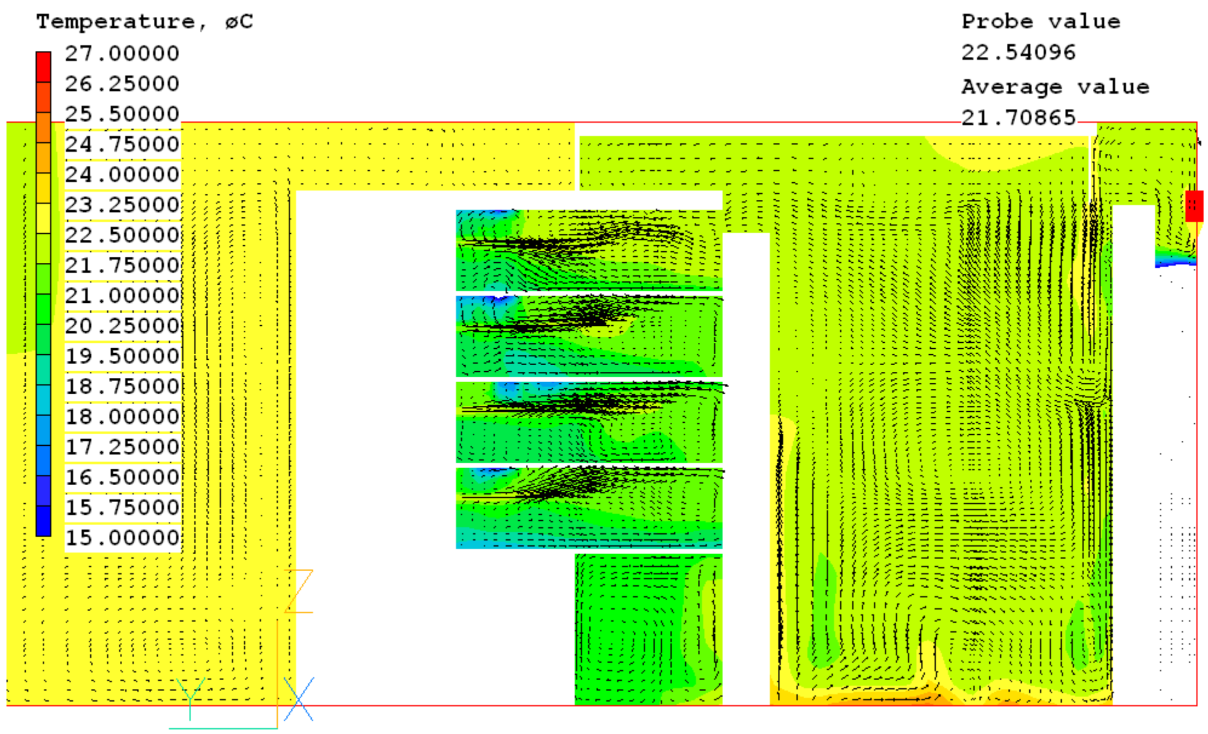
A converged solution was obtained after 2000 iterations, which took 22 hours to complete on a 3MHz PC, and 8.5 hours on an equivalent 4-processor cluster using the parallel version of PHOENICS.

Results

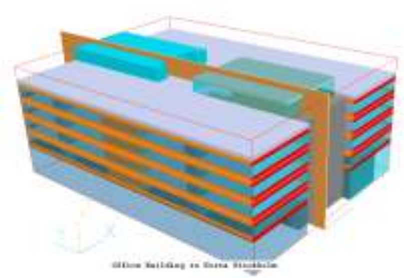
Seminar

CHAM

- Summer temperatures – X plane

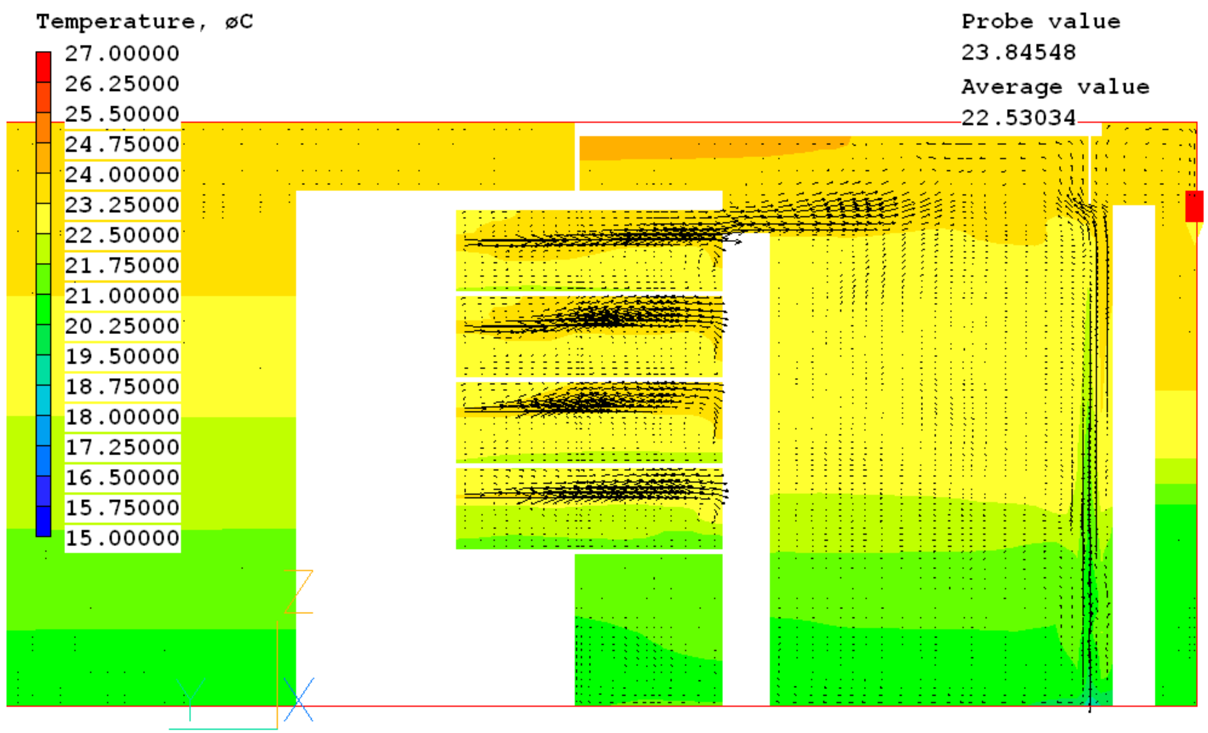


Office Building in Kista Stockholm

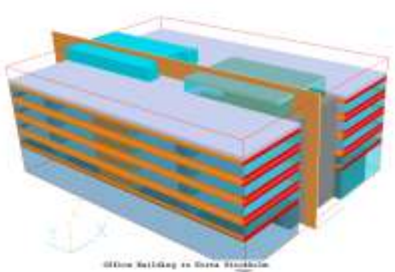


Results

- Winter temperatures – X plane



Office Bldg in Kista Stockholm (Winter)

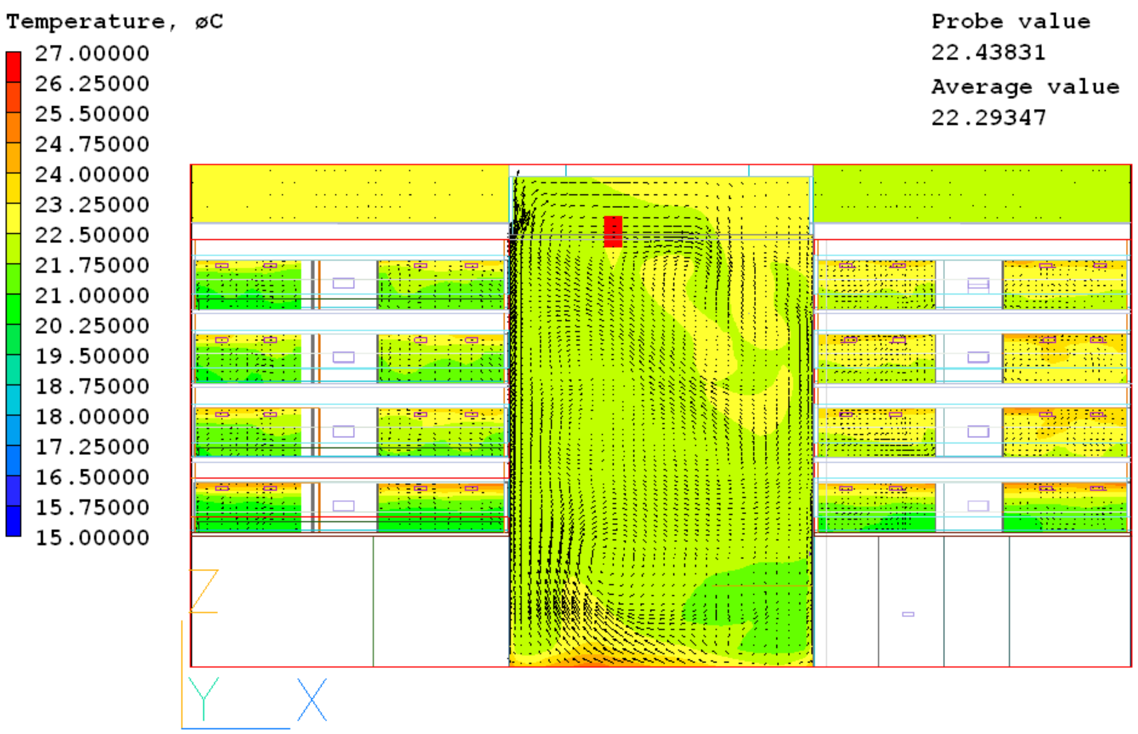


Results

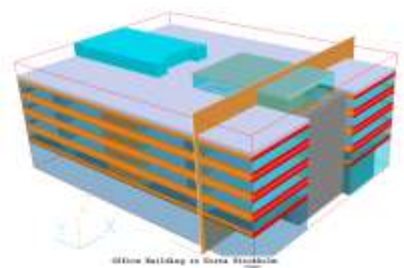
Seminar

CHAM

- Summer temperatures – Y plane



Office Building in Kista Stockholm

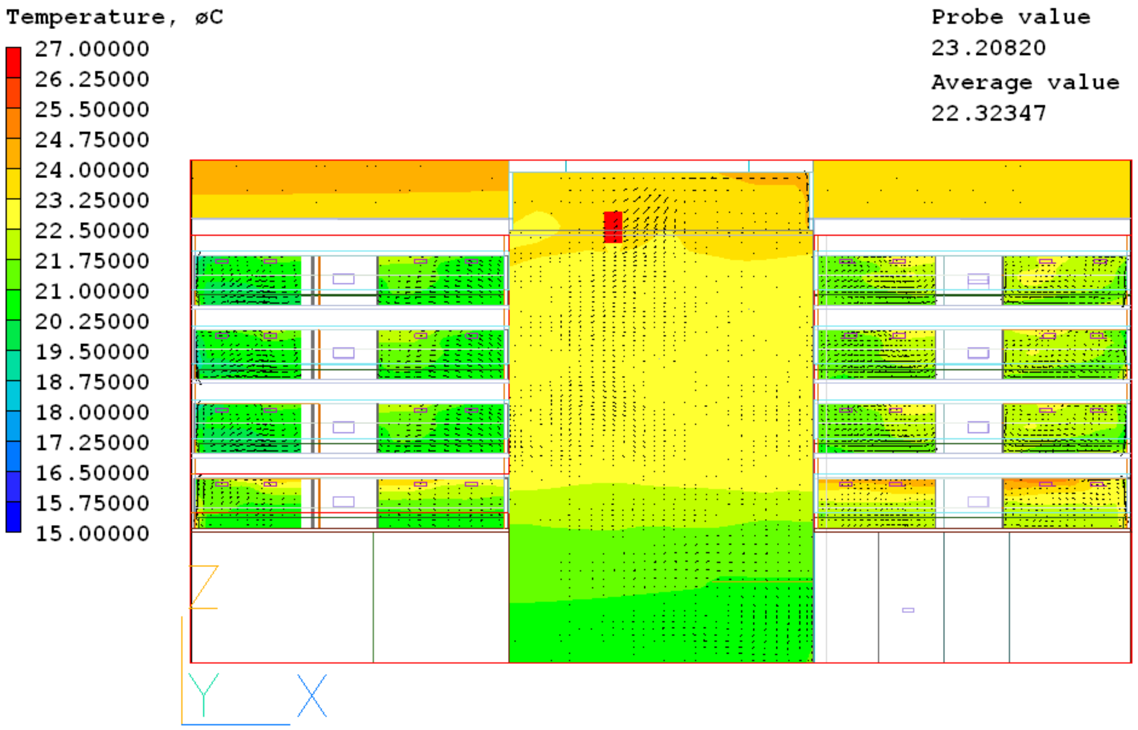


Results

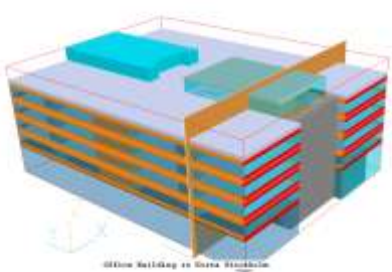
Seminar

CHAM

- Winter temperatures – Y plane

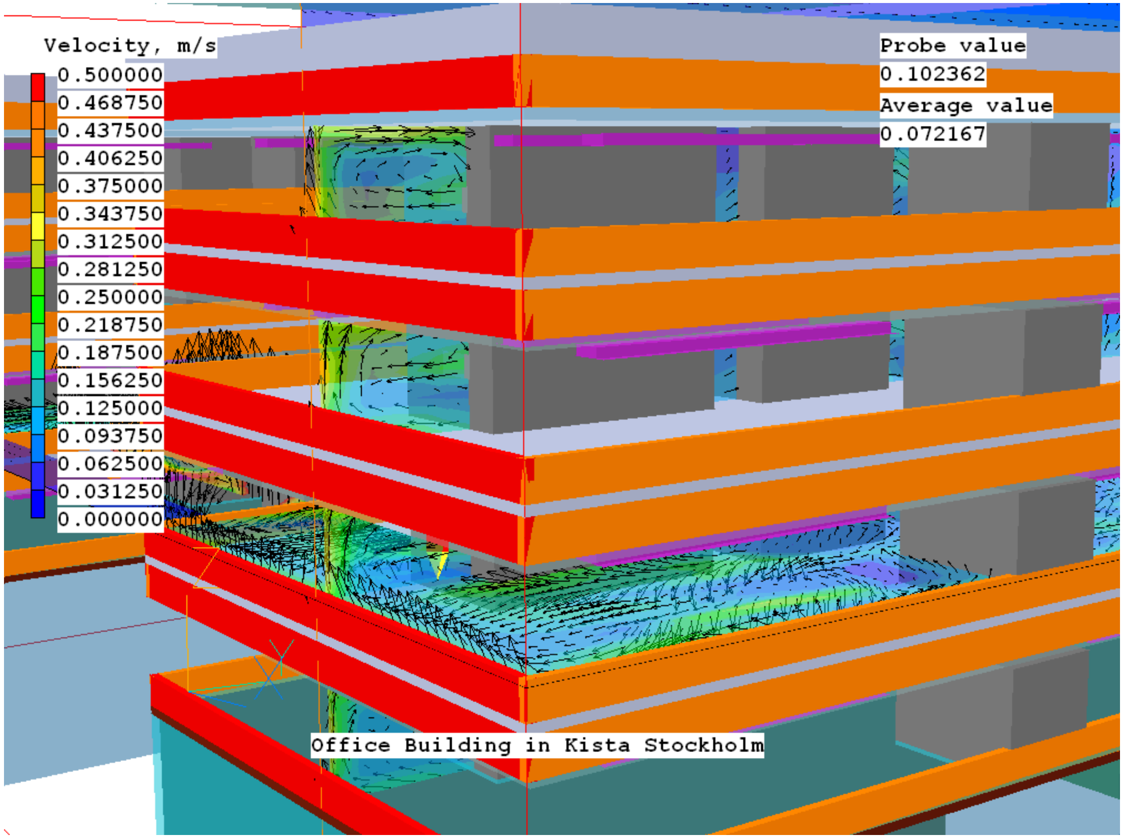


Office Bldg in Kista Stockholm (Winter)



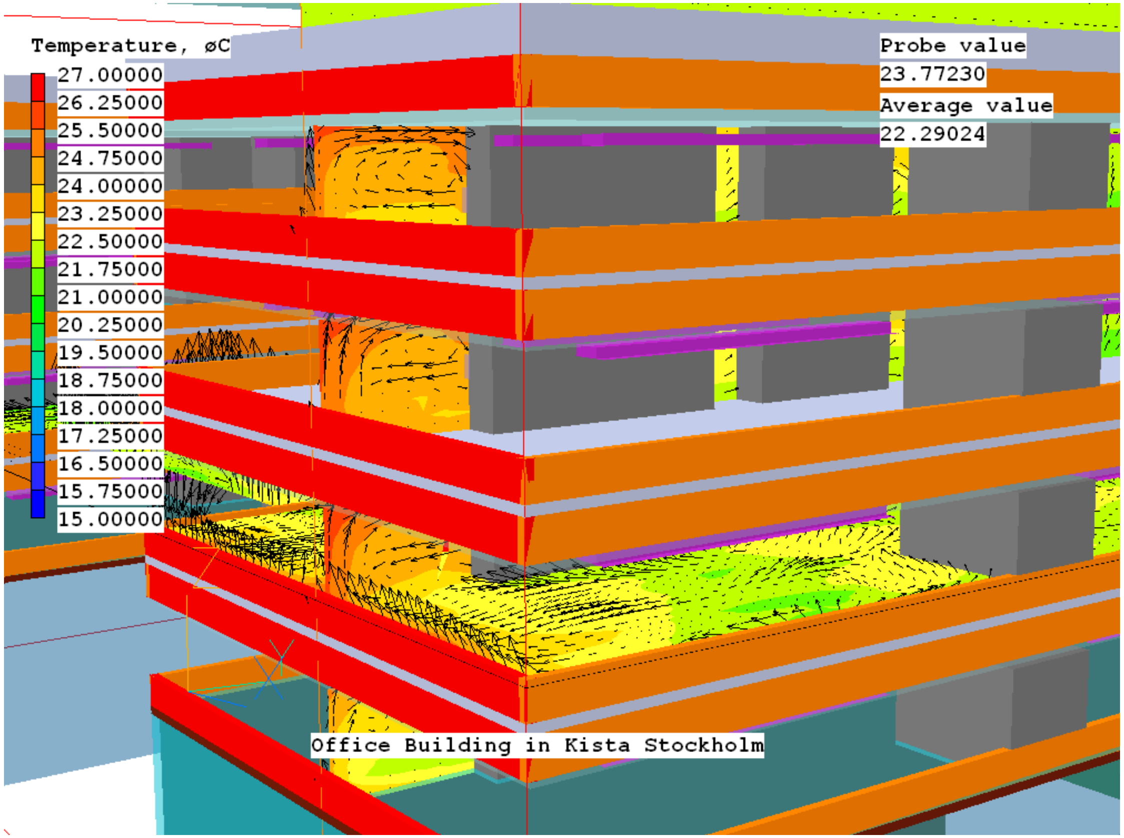
Results

- Velocities in one of the rooms



Results

- Temperatures in one of the rooms

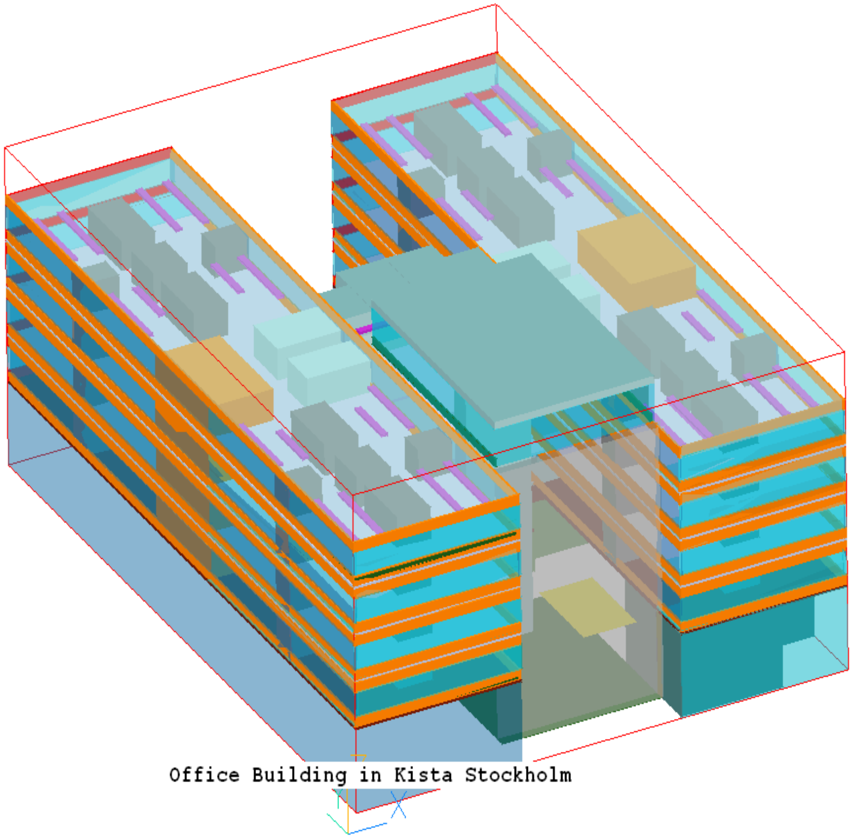


Conclusion

Seminar

CHAM

These, and more-detailed, results were supplied to support evidence from CHAM's customer to demonstrate the effectiveness of the building's HVAC design under atypical weather scenarios.

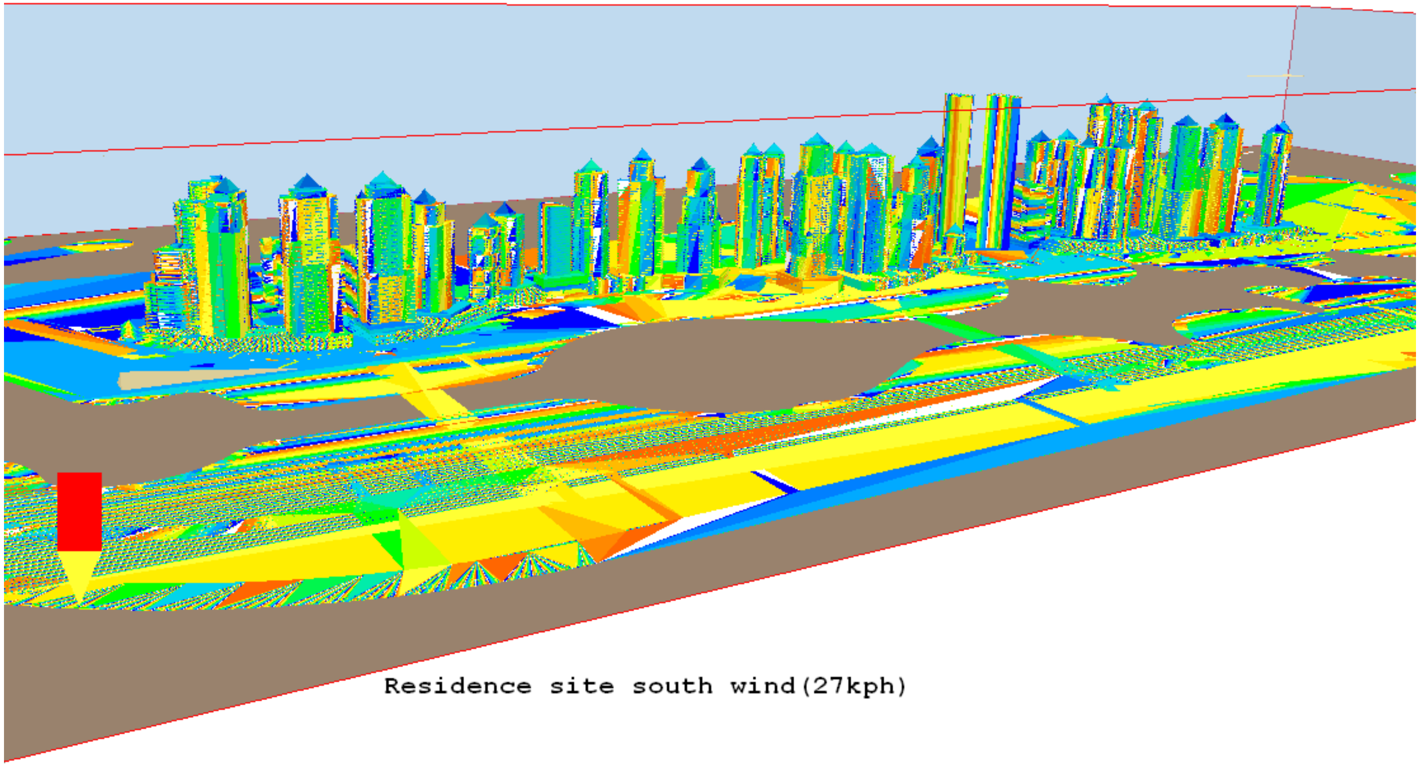


Seminar

CHAM

PHOENICS Applied to Large-scale Environmental Flows

The work concerns localised environmental conditions which could affect the occupants of the buildings as well as pedestrians.



Large-scale Environmental Flows

The objectives of this project are:

- to investigate the influence of different wind speeds and wind directions on the air flow throughout the residential area;
- to reveal any unusual wind patterns that may cause suction and up- and down-drafts that could render podium, balcony, penthouse or terraced areas at lower or upper levels dangerous to the residents.

In the past, such an investigation would have required:

- the construction of a small-scale model of the proposed complex of buildings,
- placing the model in a wind-tunnel, and
- making extensive measurements.

Nowadays, use of simulation techniques enables the same information to be obtained more swiftly, and at smaller financial cost.

Geometry and calculation domain

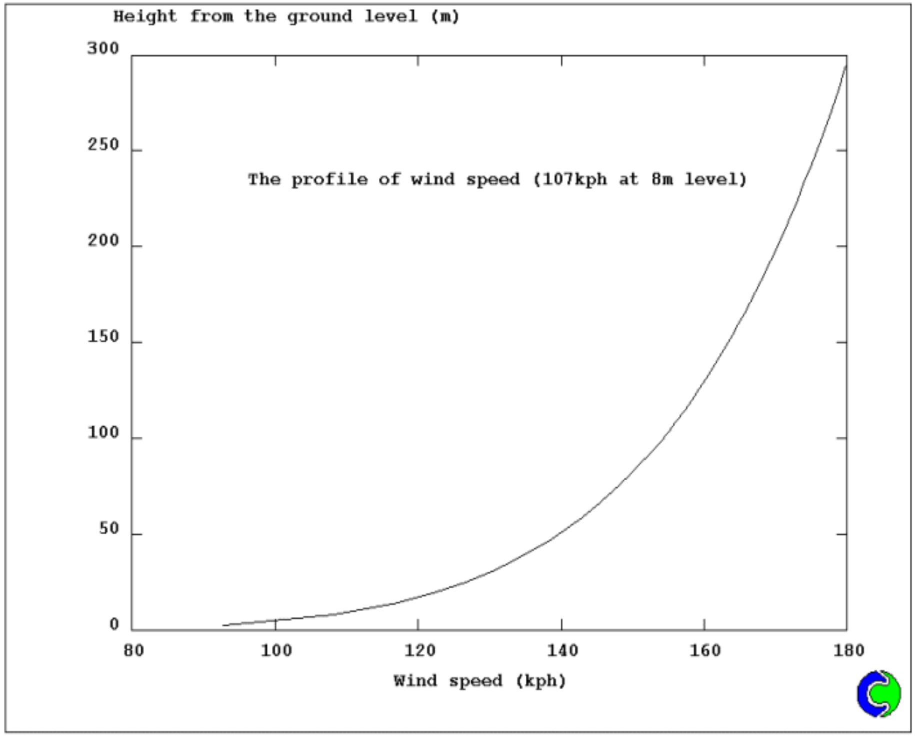
- The calculation domain covers an area of 2939m x 1300m, provided as a single geometry file, including all the buildings and surrounding areas.
- The height of 302m provides about 100m open space above the tallest building.

Physical modelling

- Three-dimensional conservation equations are solved for mass continuity and momentum.
- The flow is steady.
- The Cartesian co-ordinate system is employed. A non-uniform mesh distribution is adopted with finer meshes assigned around the buildings. The grid uses 208 x 167 X 46 cells.
- Ground friction is considered.
- The turbulence is represented by the LVEL turbulence model

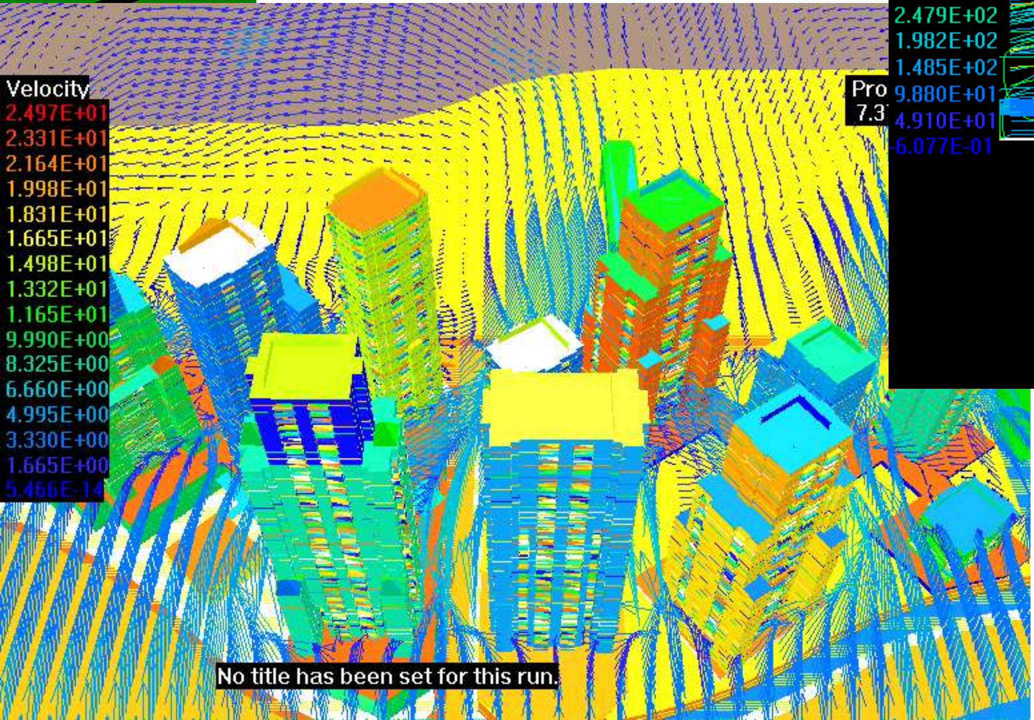
Boundary conditions

- A wind profile of $U^{1/7}$ with the measured wind speed at a height of 8m is employed at the boundaries where the wind enters the domain.
- In-Form is used to set the boundary layer profile.



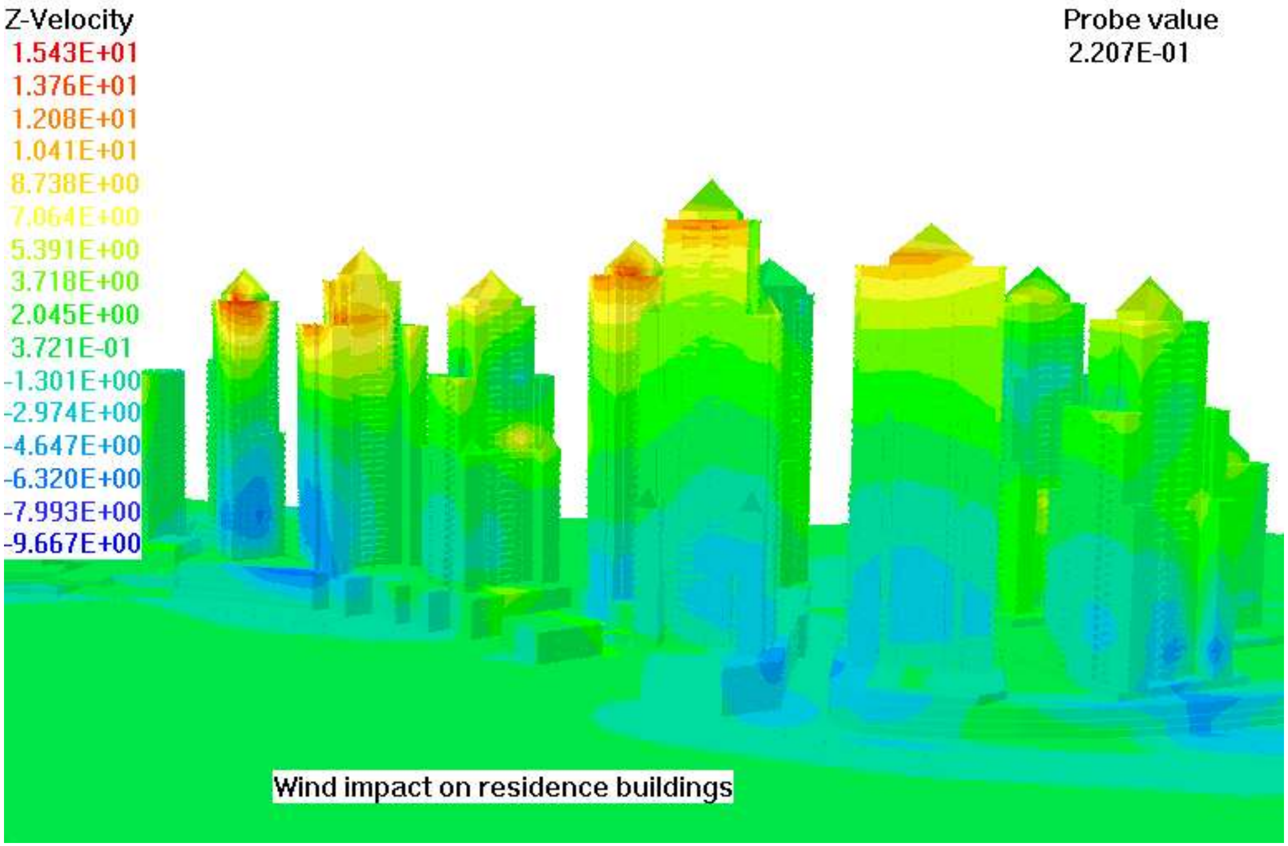
Large-scale Environmental Flows

The results show that the predicted localised wind speed increases as the incoming wind speed increases and as the height from the ground increases.



The maximum wind speed could reach over 200 kph.

Large-scale Environmental Flows



In-Form was used to deduce the velocity in kph from the standard PHOENICS m/s.

Large-scale Environmental Flows - Problems

The geometry was supplied by the client as a single STL file.

Unfortunately the CAD packages used by architects do not necessarily guarantee that the facets are consistent with each other in respect of inward- and out-ward-looking direction or define closed volumes.

PHOENICS requires that facets should have a direction sense in order that it can determine on which side is the fluid and on which the solid; and of course facets which share an edge should be in agreement on this matter.

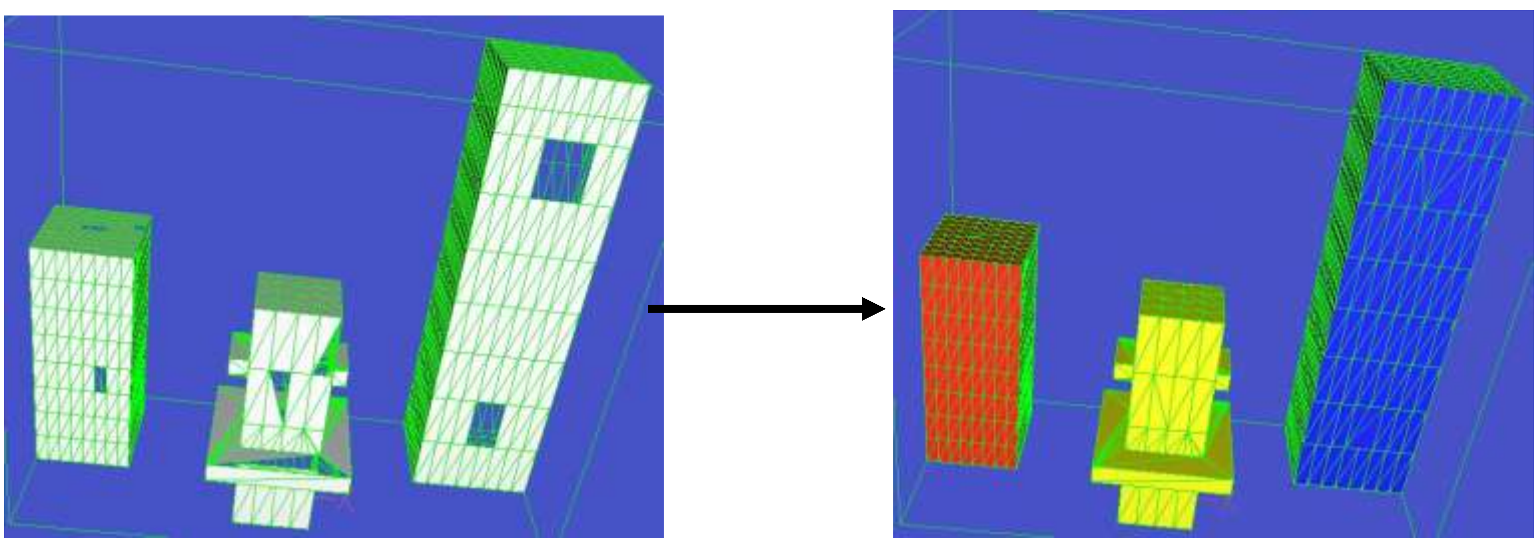
A further requirement is that the facets defining an object should, taken together, form a complete closed surface, such as is possessed by every solid body.

Large-scale Environmental Flows - Problems

The file supplied suffered from all these defects.

- some facets were facing the wrong way. Parts of the buildings could not be detected.
- there were holes in solid bodies, allowing fluid to leak in and solid to leak out

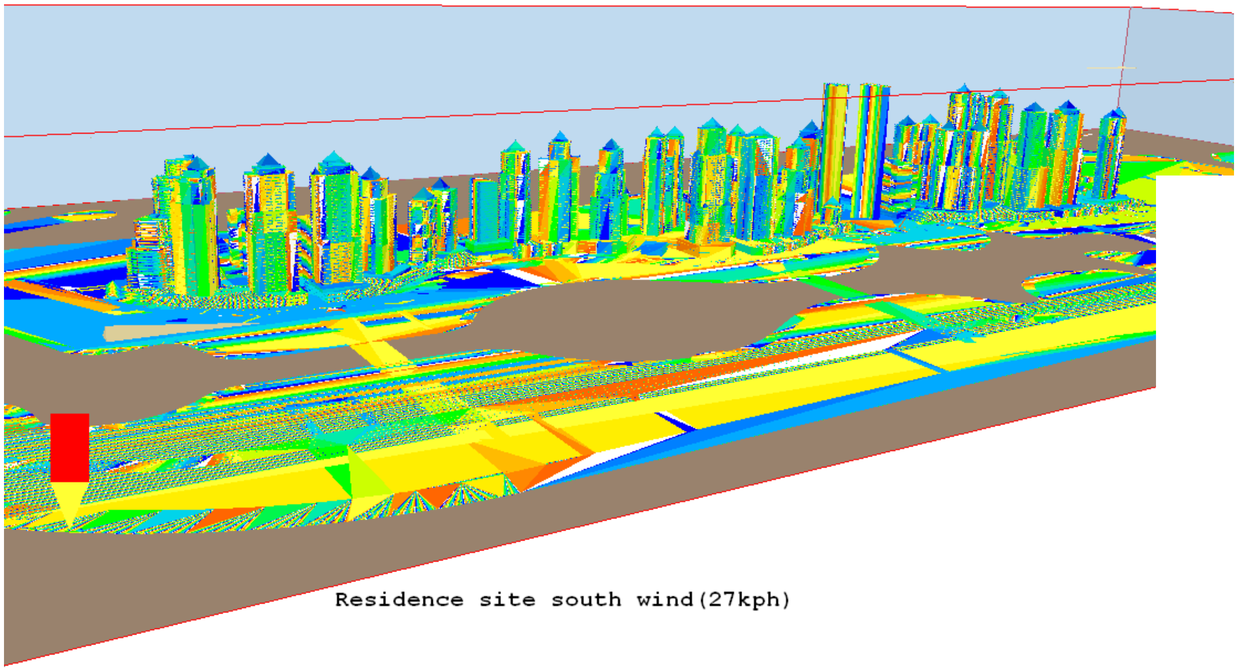
The solution was to create a program to 'fix' the STL file. **FacetFix** is now supplied as a standard utility to repair defective STL files, enforce consistency, add facets, and create new .dat files needed by PHOENICS.

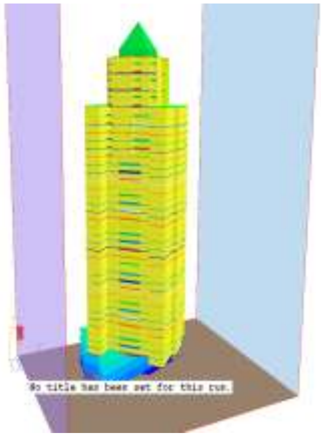


Large-scale Environmental Flows - Problems

It can also do the same for defective .dat files.

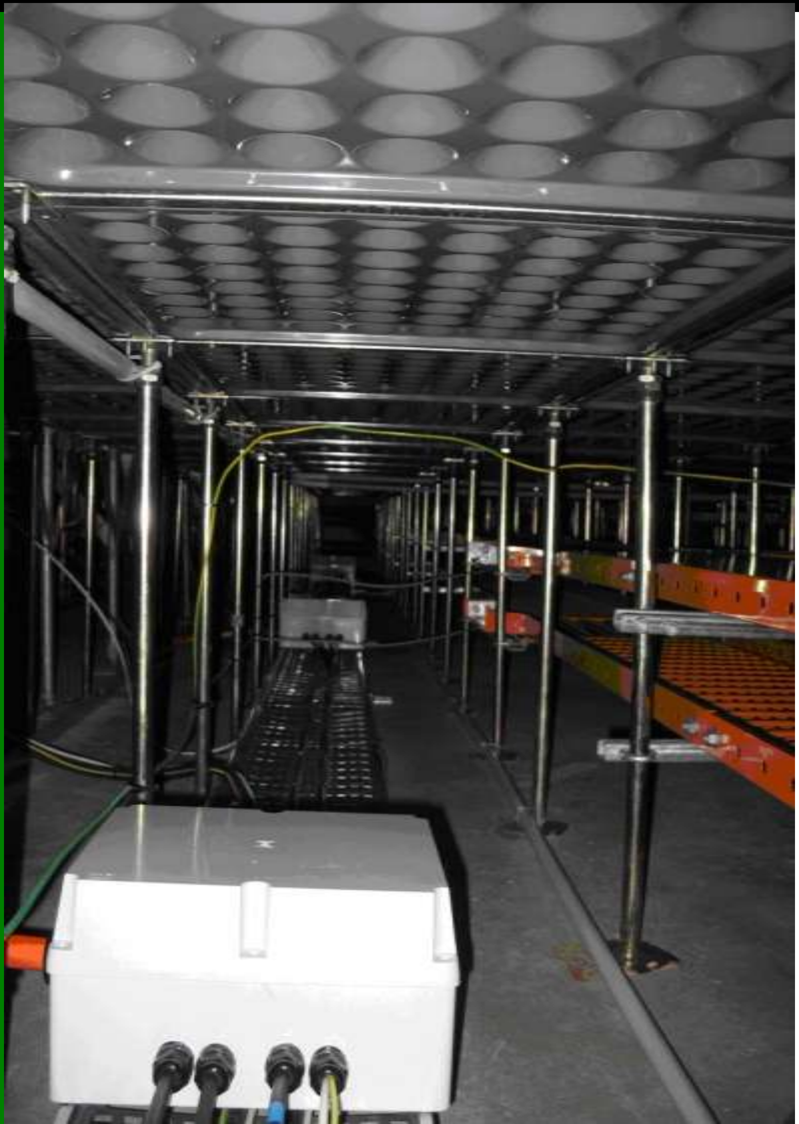
It can extract facets from a specified volume and make them into a solid body. This allows a single building to be extracted from the complex.



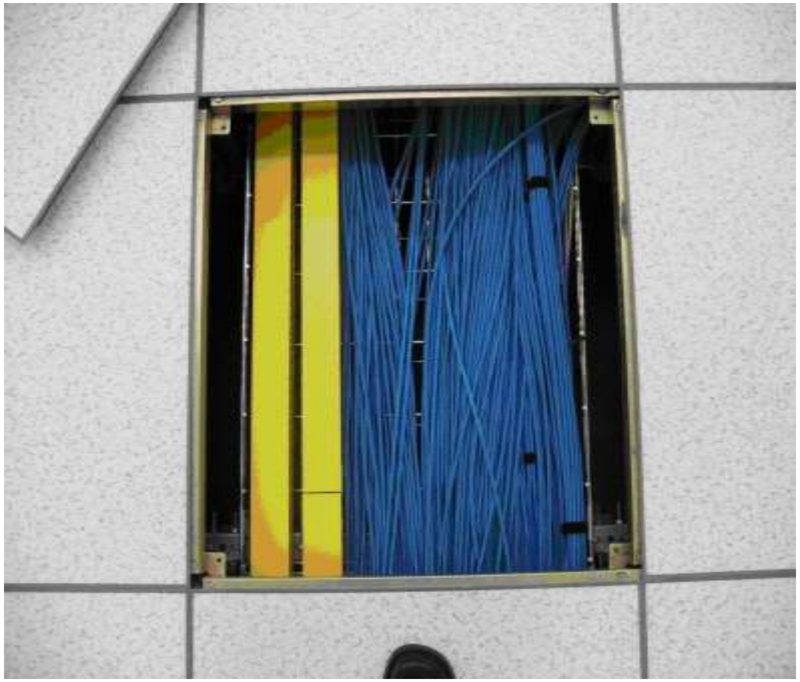


Data Centre Cooling Control

CHAM



Data and graphics courtesy of
Stephen Grubitts & Associates



Data Centre Cooling Control
Design issues

- **Heat Rejection Sources**

  IT racks with blade servers: 2 ~ 7 kW/rack

- **Cooling System (typical)**

  Under floor supply cooling air from CRAC units

- **ASHRAE TC 9.9: Temperature Control for Class 1 Data Centres**

  Allowable: 15 ~ 32 ^{o}C

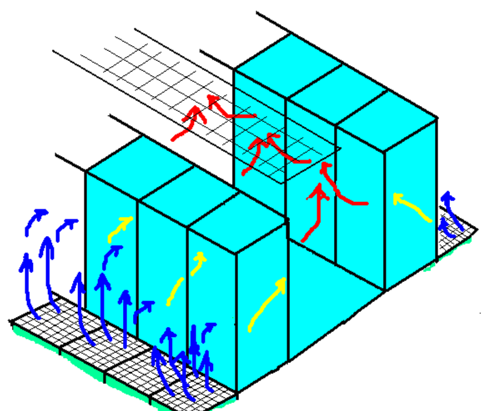
  Recommended: 20 ~ 25 ^{o}C

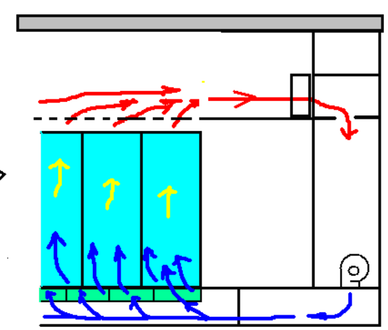
- **Cabling Conditions under Floor Plenum & above Racks**

Data Centre Cooling Control

CHAM

Cooling air → floor grilles

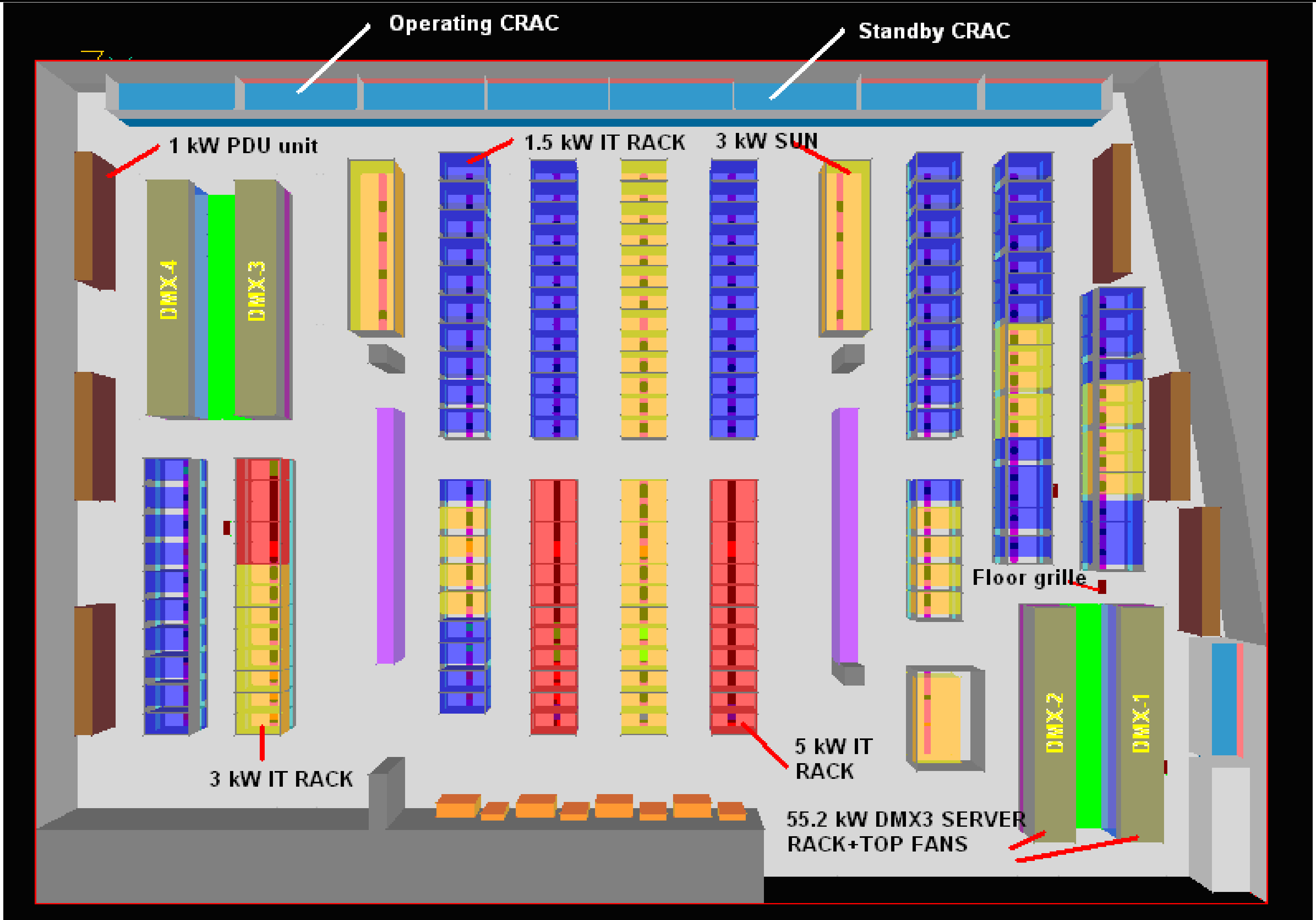




Data Centre Cooling Control
CFD model

Seminar

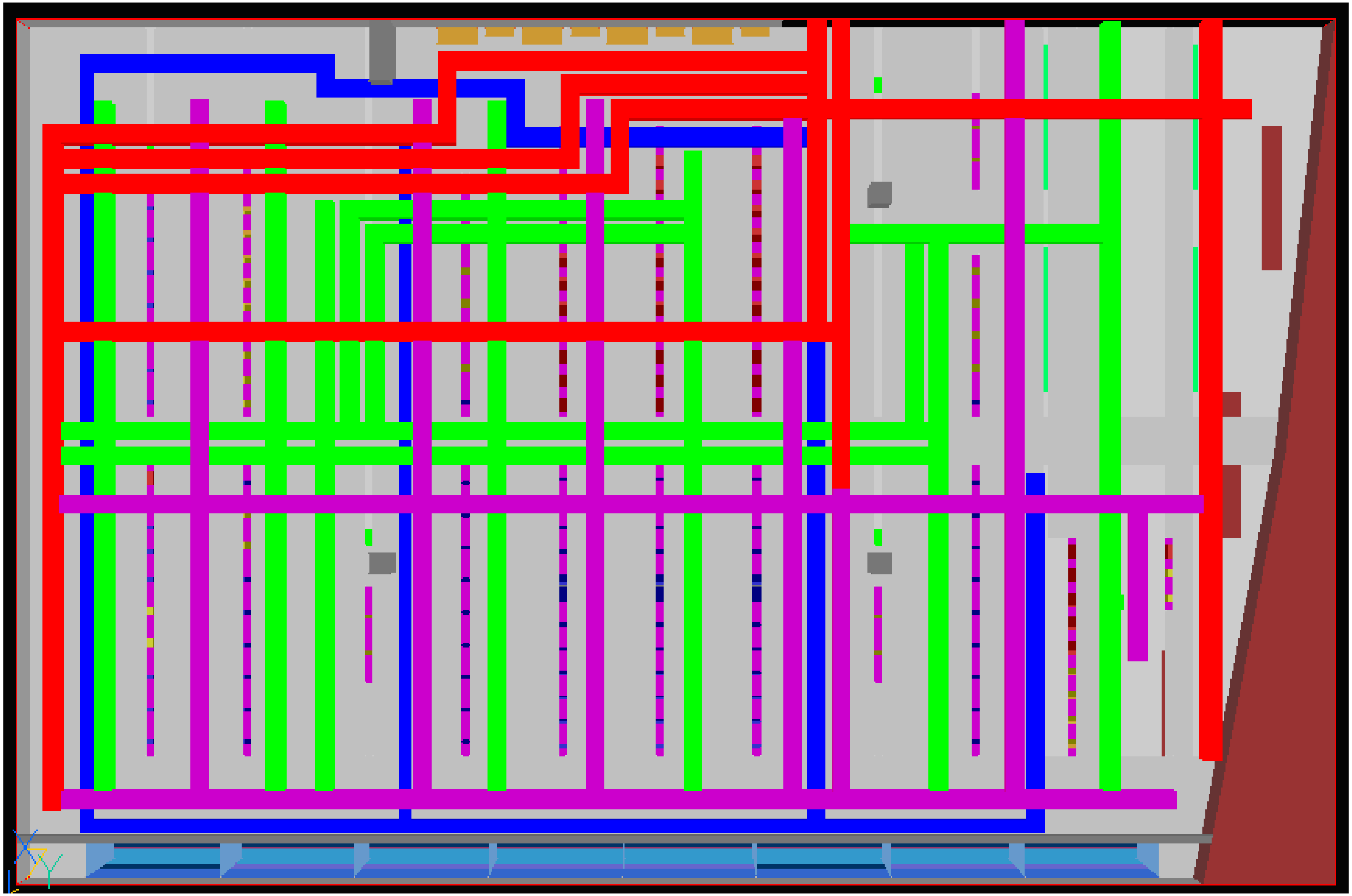
CHAM



Data Centre Cooling Control
CFD model - cabling

Seminar

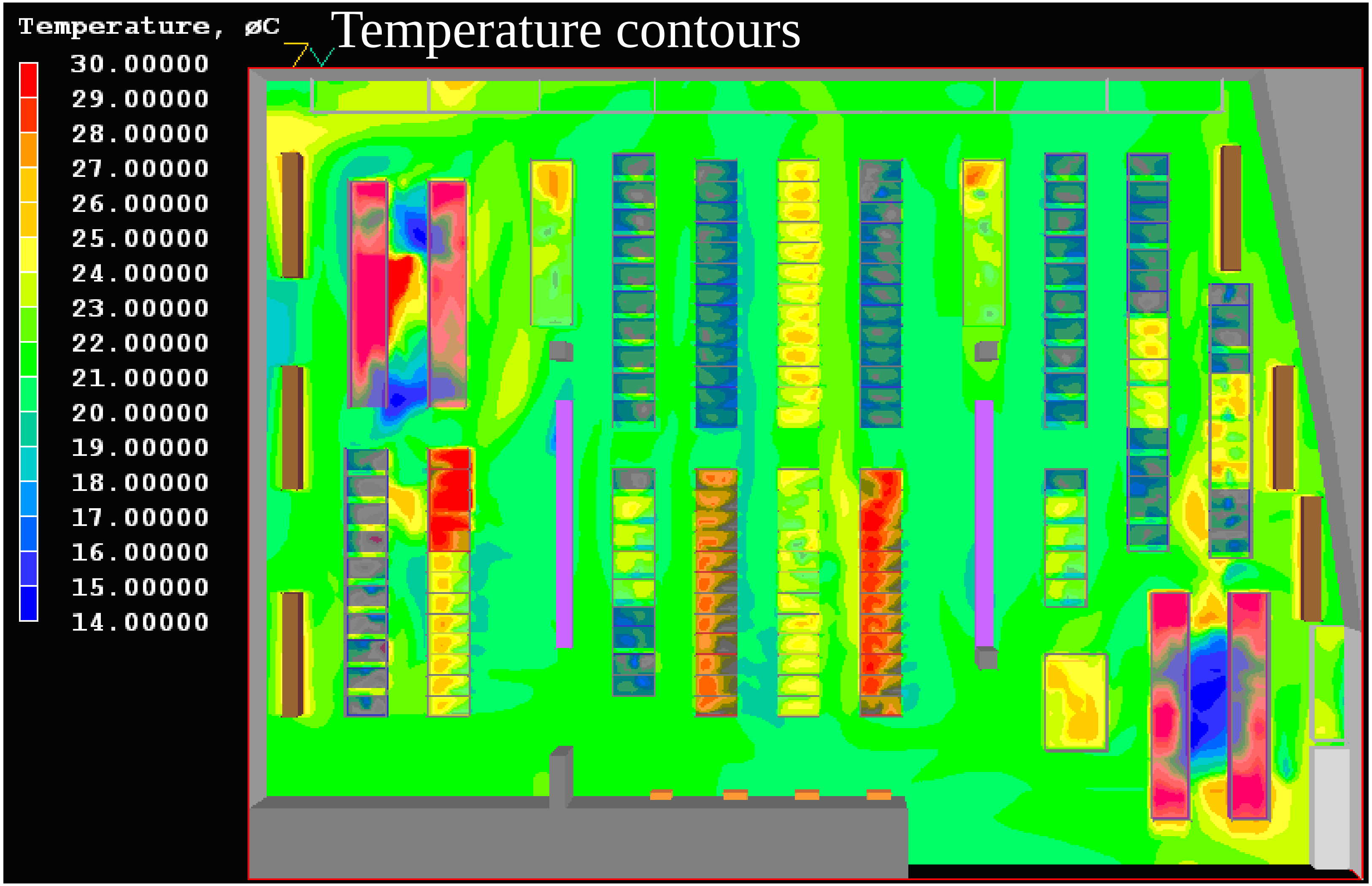
CHAM



Data Centre Cooling Control Results

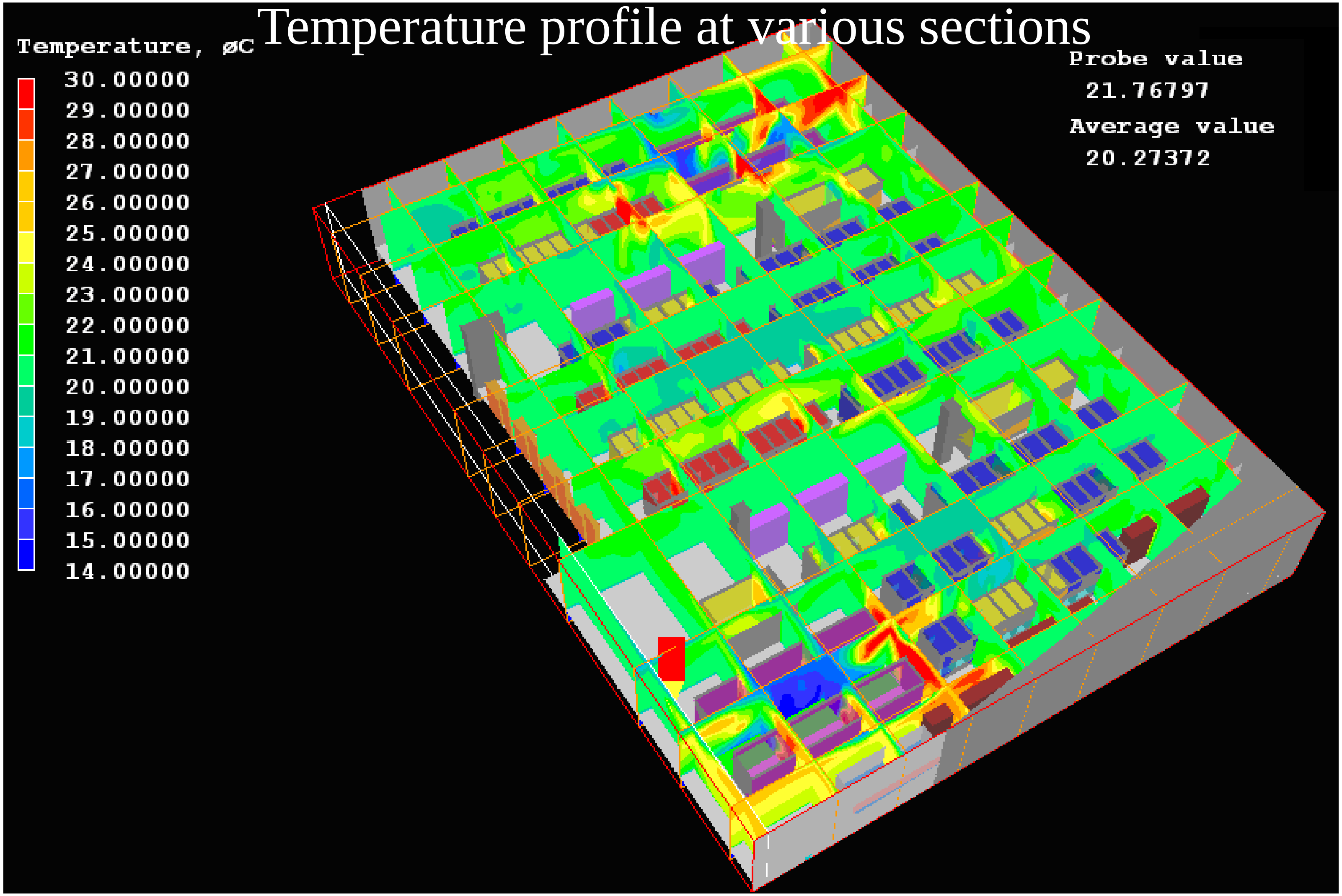
Seminar

CHAM



Data Centre Cooling Control Results

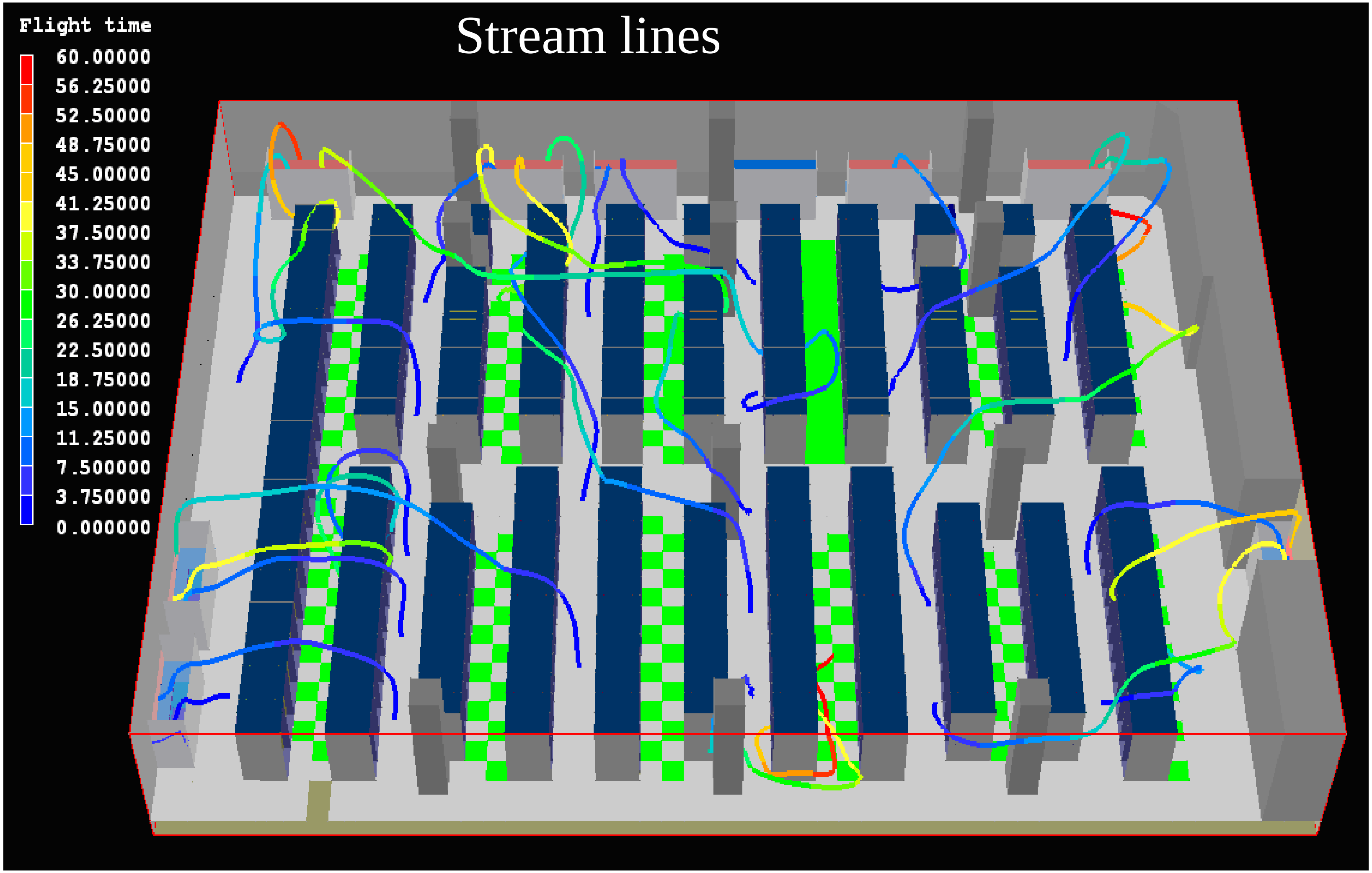
Temperature profile at various sections



Data Centre Cooling Control Results

Seminar

CHAM



Urban City Planning – Wellington, New Zealand

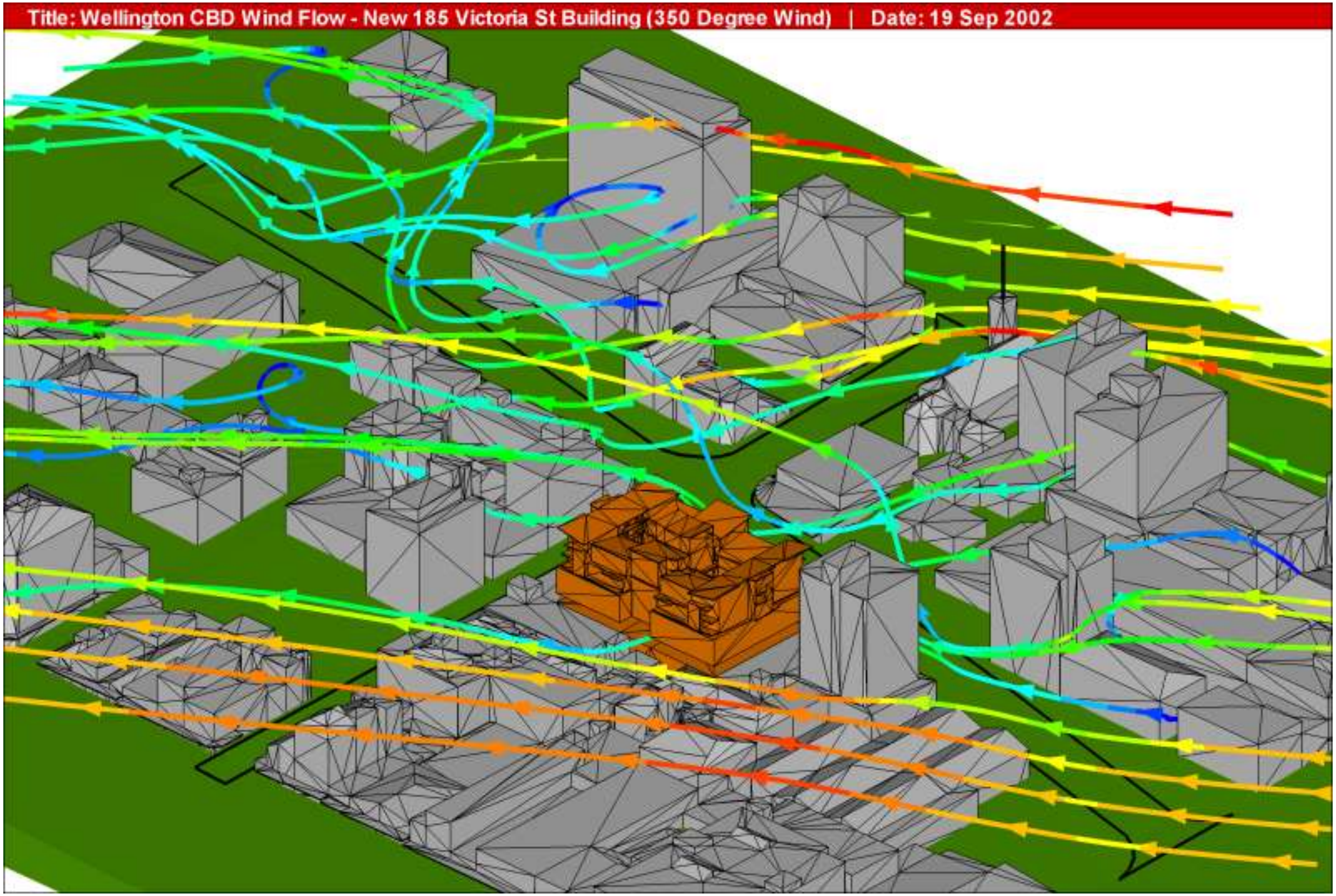
Seminar

CHAM

Urban City Planning – Wellington, New Zealand

Seminar

CHAM

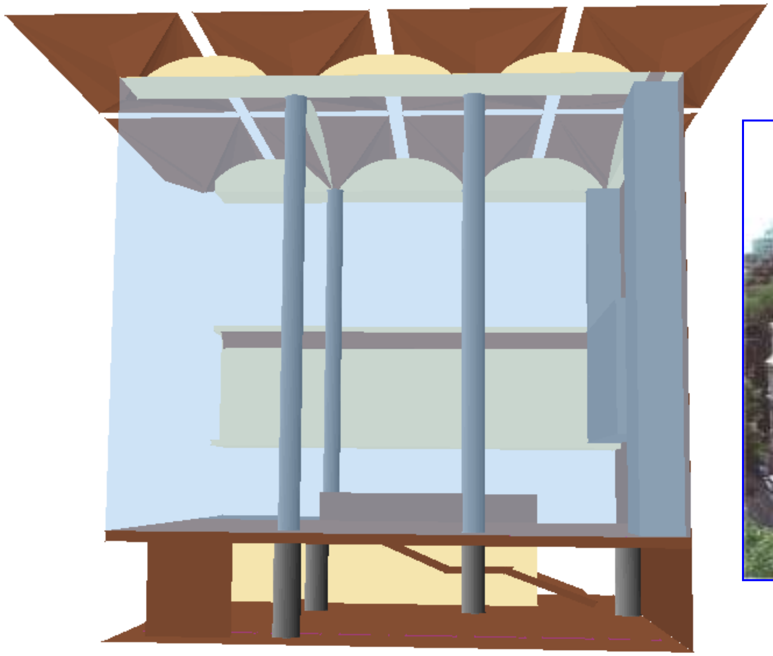


Conclusions

- The results from the CFD modelling provide a detailed analysis of the local wind speeds and highlight the impact upon pedestrian comfort and safety.
- In this way the building form can be changed early in the design stage process and enable urban planners to develop safe, yet commercially viable solutions.

The CFD analysis of airflow and temperature distribution in the atrium of an art gallery

The atrium of the new development is four levels high extending from the mezzanine leve through level 1, level 2 and level 3 to the roof. The atrium sits above the main entry at ground floor level and it is surrounded by 'platforms' at each level overlooking the central void space.



Atrium

The objective of the project

The objectives of this project are:

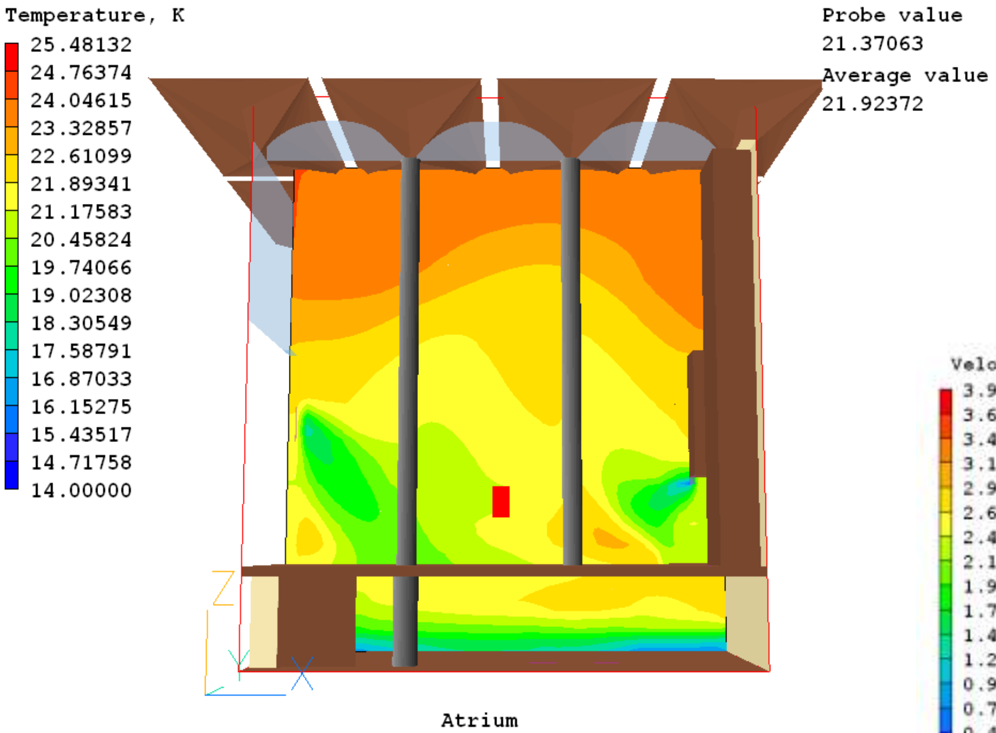
1. To assess the effects of air conditioning performance of various population in the Atrium and galleries, and the impact of one upon the other, using defined ducting and air conditioning configuration and architectural structure;
2. To investigate the effects of the grille arrangement in the Atrium on the temperature distribution at the head high above the mezzanine floor.

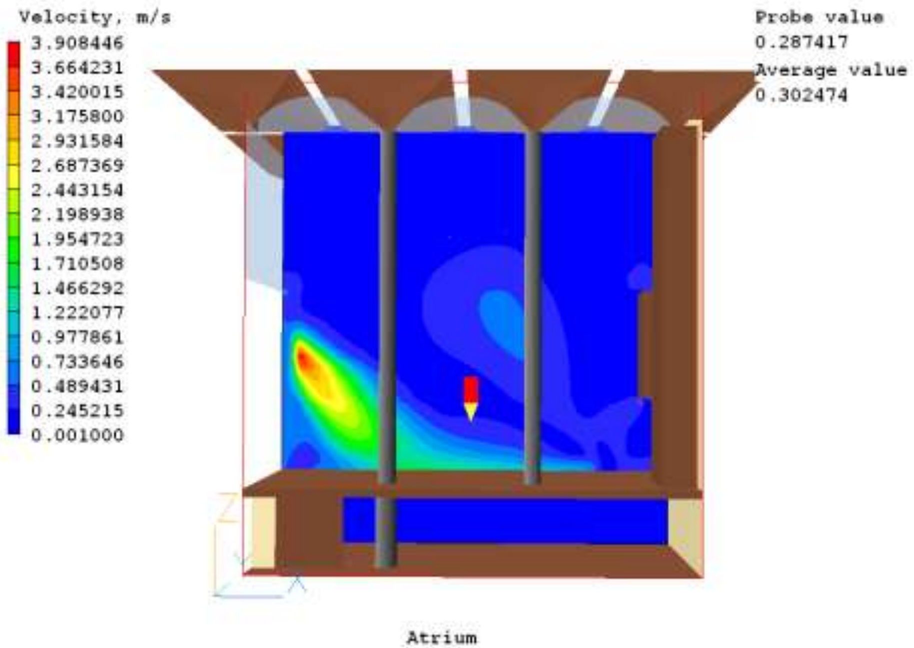
Problem specification

- The total supply air to the atrium is 12.0 m3/s respectively; (2.5 at GL; 6.5 at MZ; 3.0 at L1)
- The temperature of the supply air is 14 C;
- The total return air is 12.0 m3/s; (1.0 at GL;3.0 at MZ;4.0 at L1; 4.0 at L3);
- A large grille is mounted on the North wall of Atrium to provide the supply air with its exit velocity of 6 m/s;
- The lighting heat load used is 45w/m2 at high level;
- The environmental temperature is 28C;
- The occupancy load is 30 people at the ground floor and 700 people in the atrium.

Temperature and velocity contours

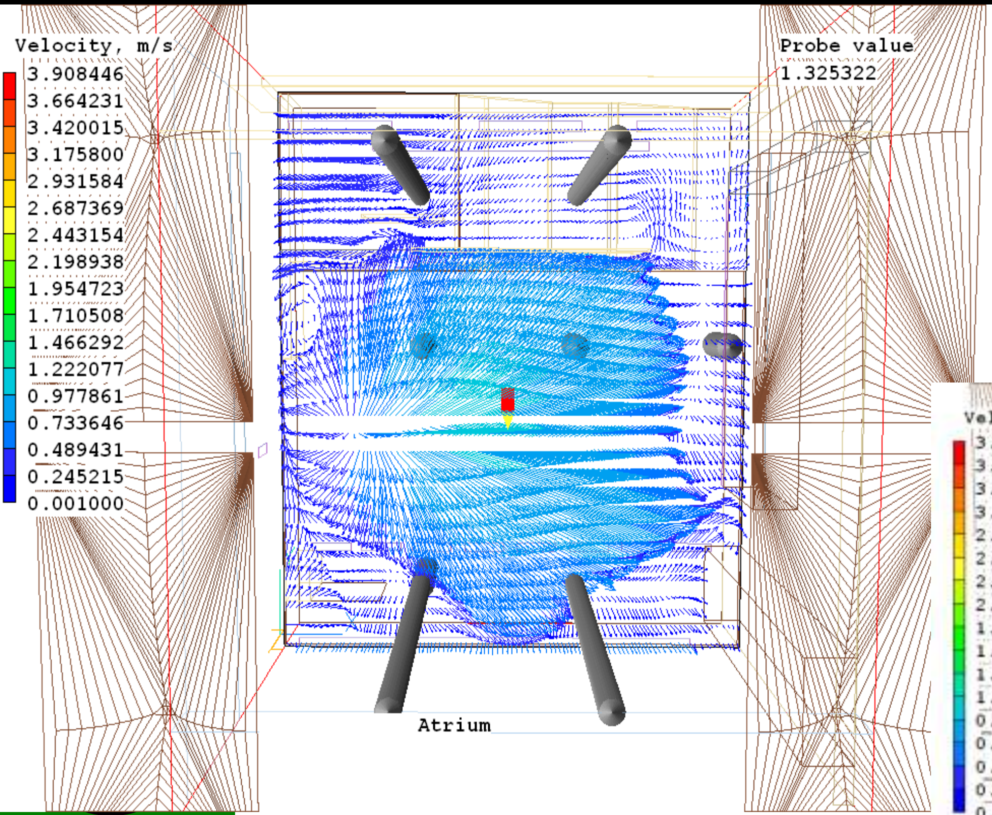
Seminar

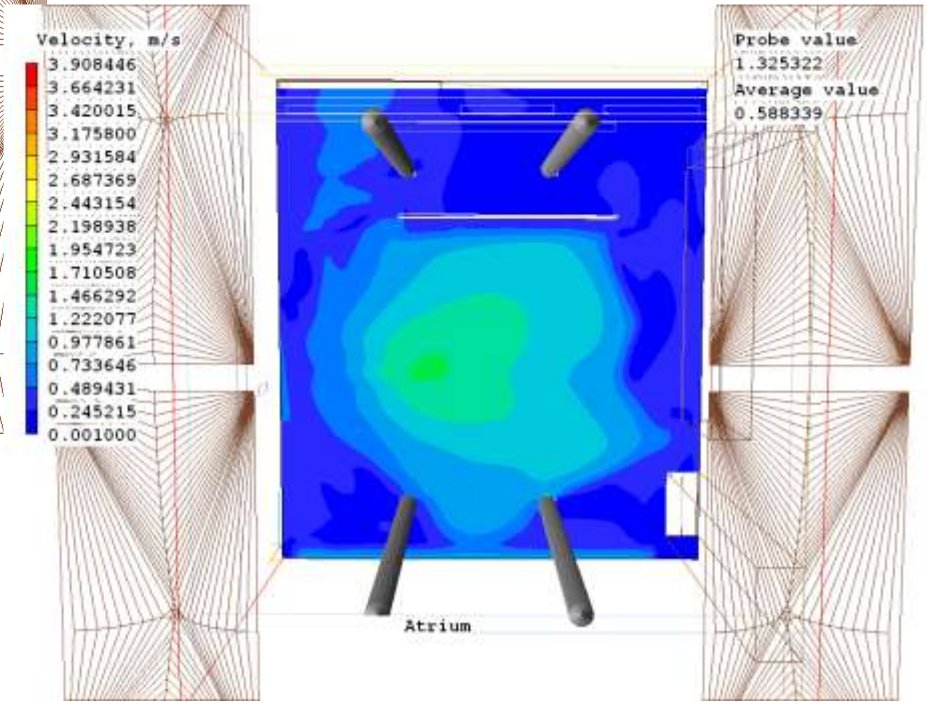




Velocity field near the floor level

Seminar

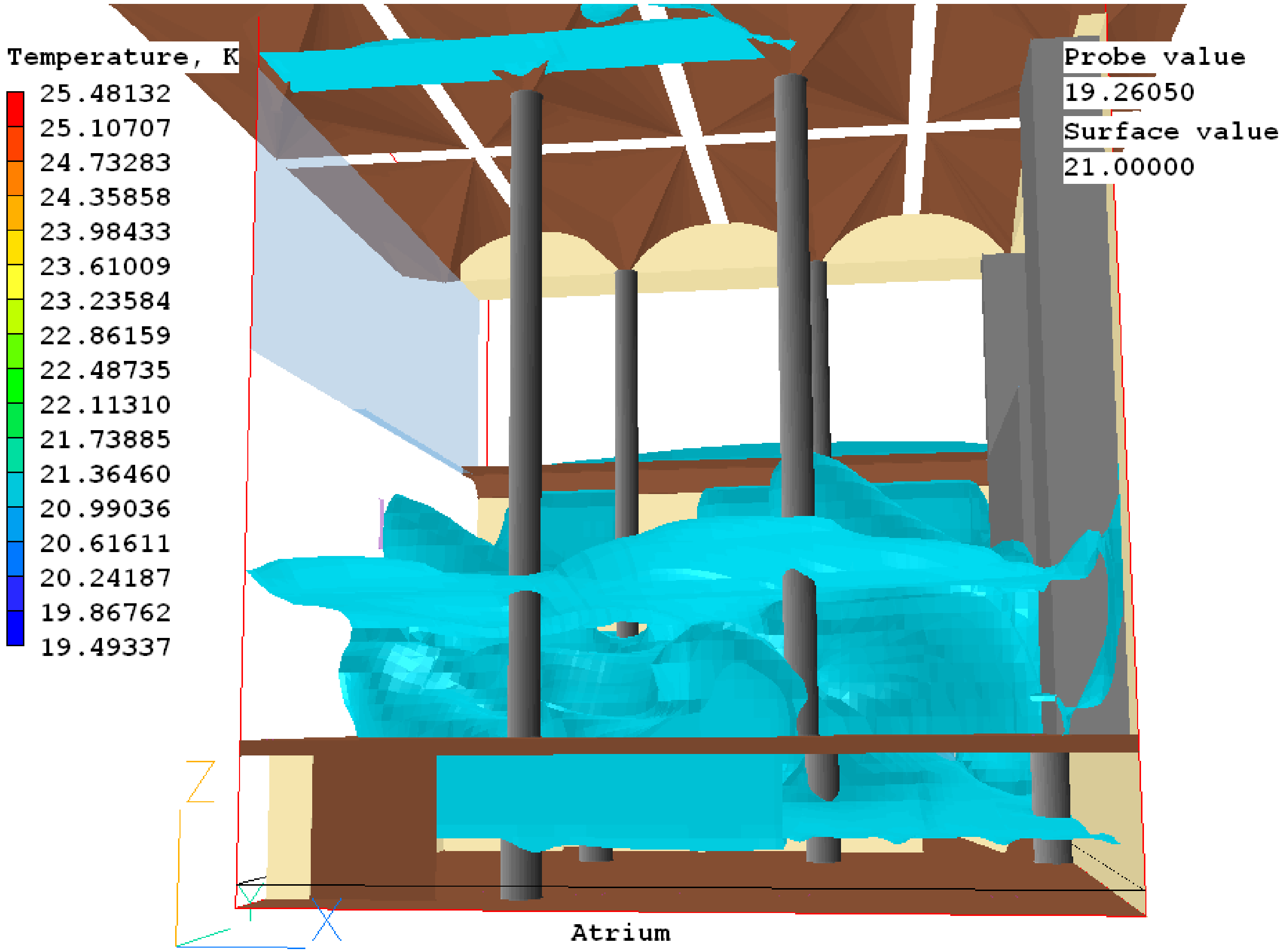




Iso-surface of 21 C

Seminar

CHAM



Findings

As expected, stratification will occur due to the height of the Atrium. Temperatures on the L2 bridge are likely to be 3 to 4 degrees hotter than at Mezzanine level.

During hot summer weather, and whenever a large crowd is present during warmer weather, the study confirmed the need to operate the Atrium smoke/relief fans to exhaust the accumulation of hot air rising to the upper level of the entire building, that is in the L2 main gallery as well as in the Atrium.

Large grille on the North wall of Atrium essential to avoid considerable temperature variation across the Mezzanine floor area.

Moorilla Art Gallery

Data and graphics courtesy of Advanced Environmental Pty Ltd

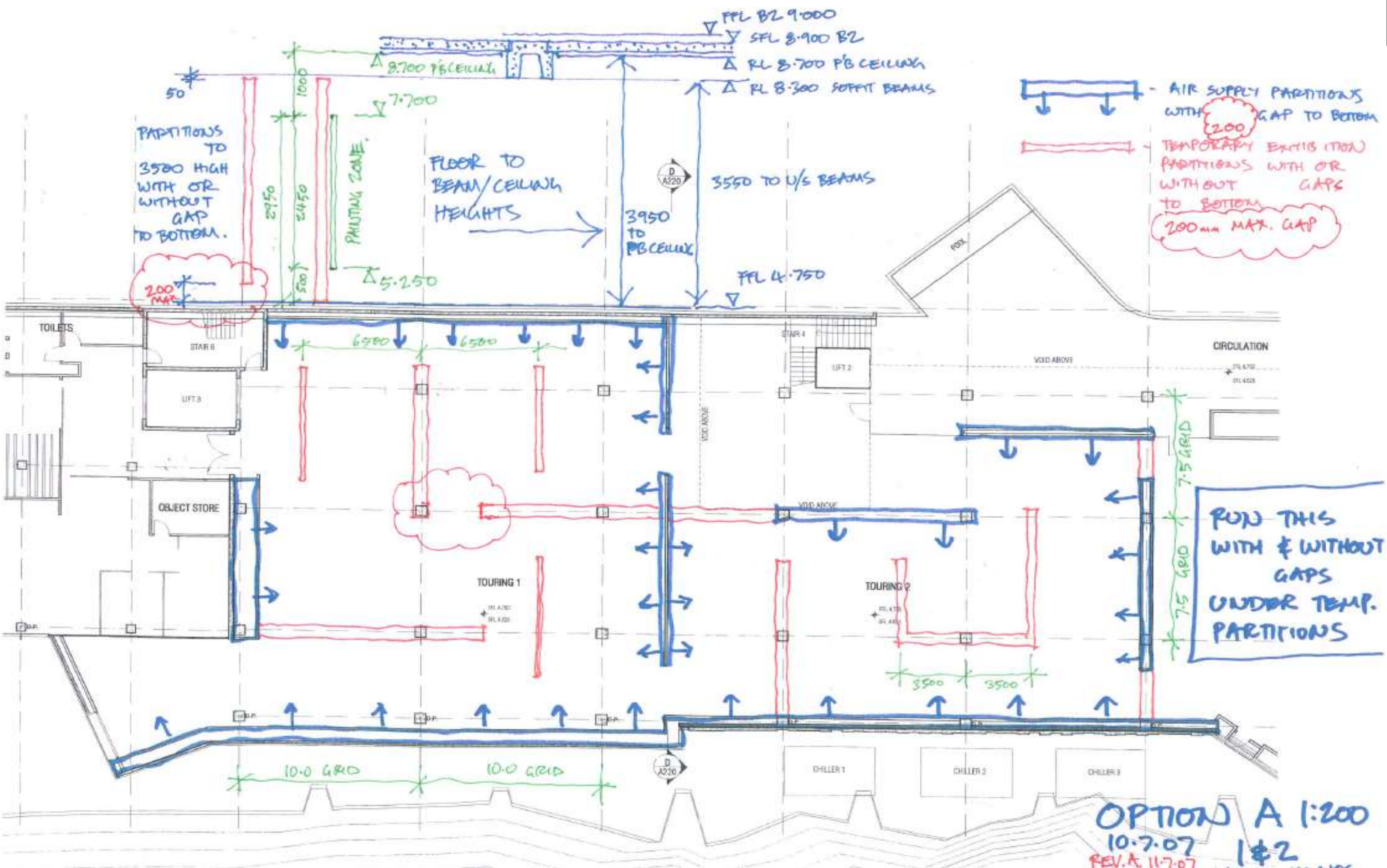
Seminar

CHAM

- Private gallery in Tasmania
- International touring gallery requiring AAA conditions
  - Temp difference 2°C across the painting zone
  - Restriction of air velocities to less than 0.35 m/s
- Displacement Ventilation, with different supply options

Moorilla Estate

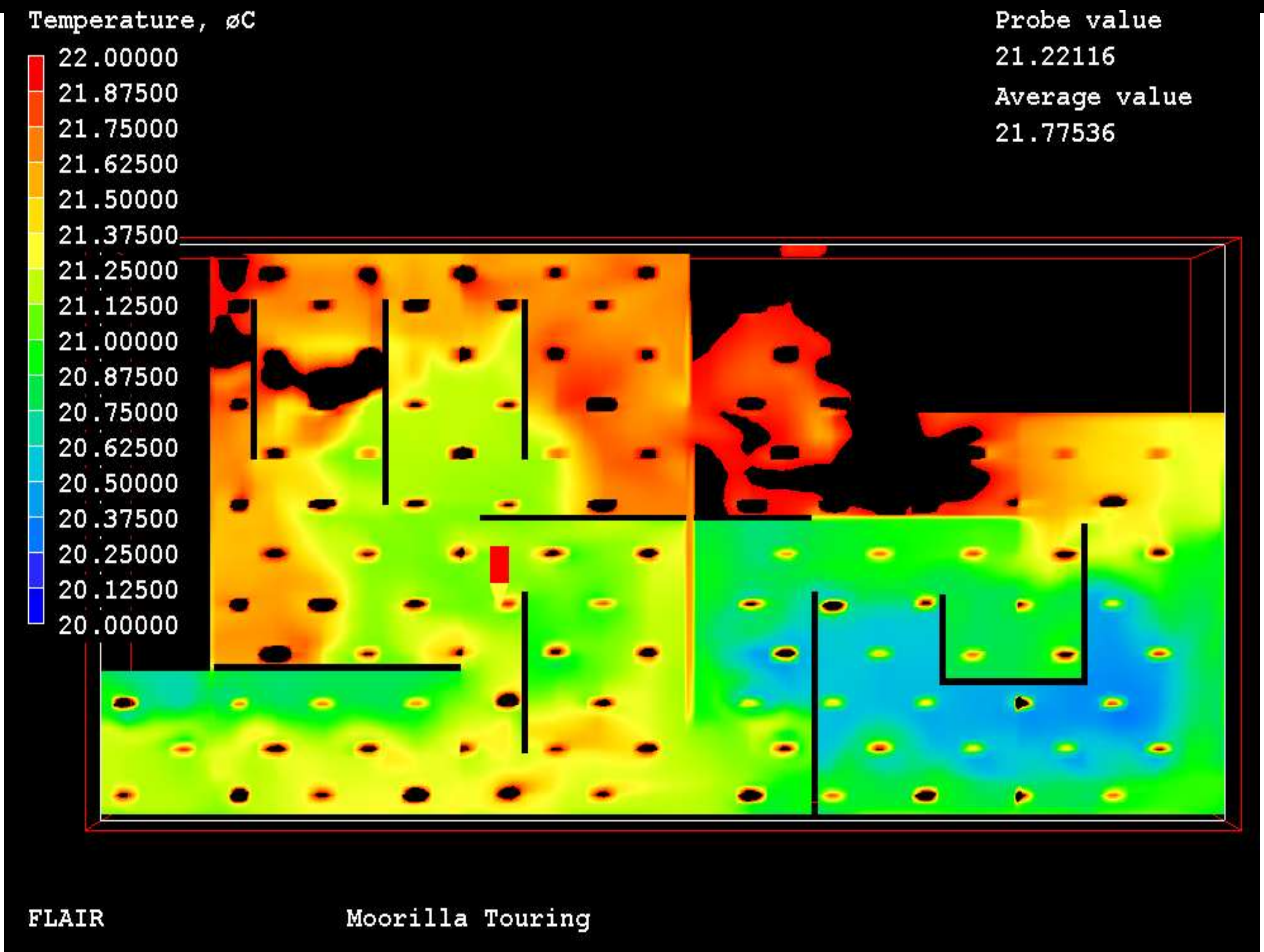
Moorilla Art Gallery



The temperature profile at the top of the painting zone

Seminar

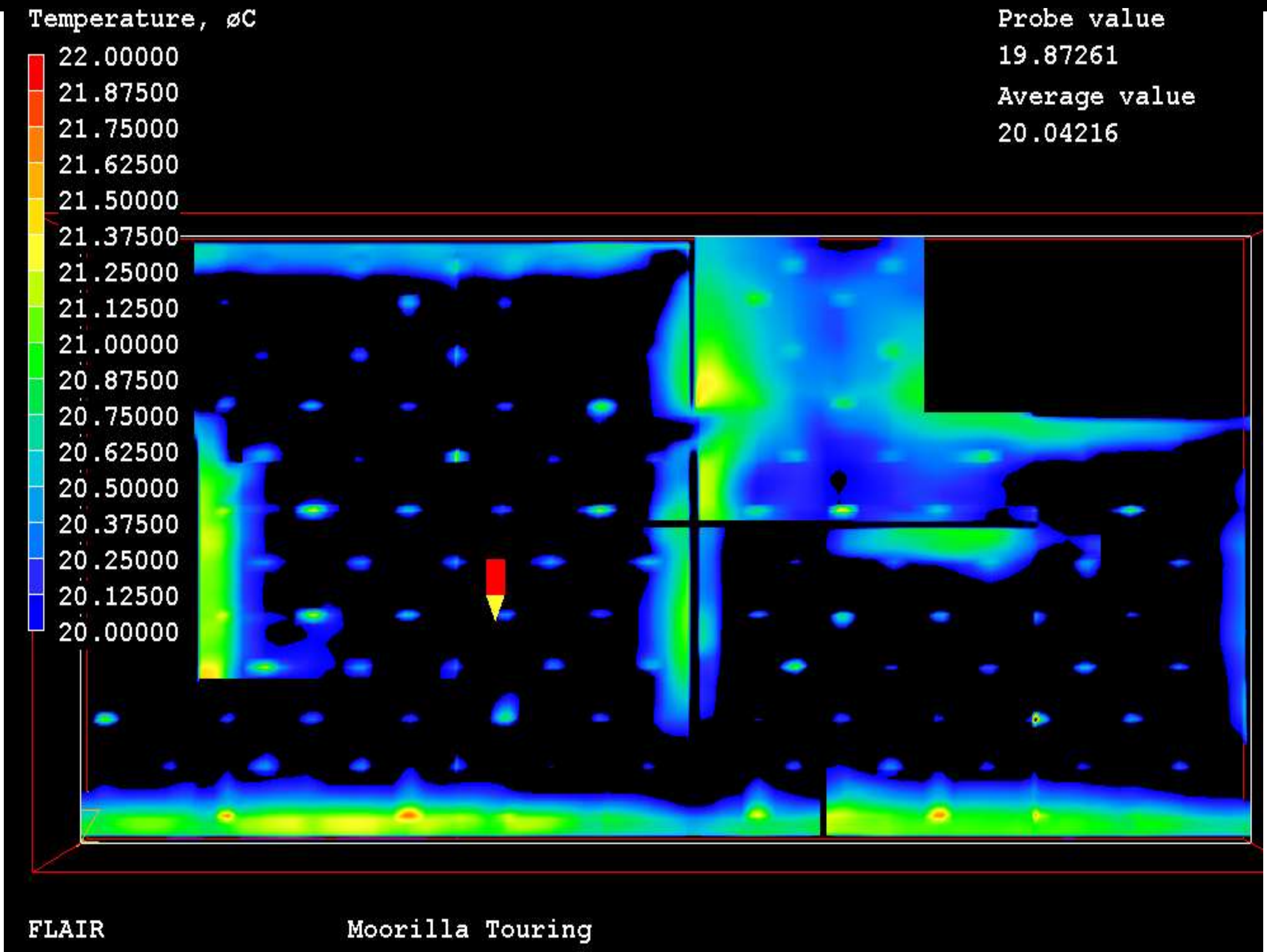
CHAM



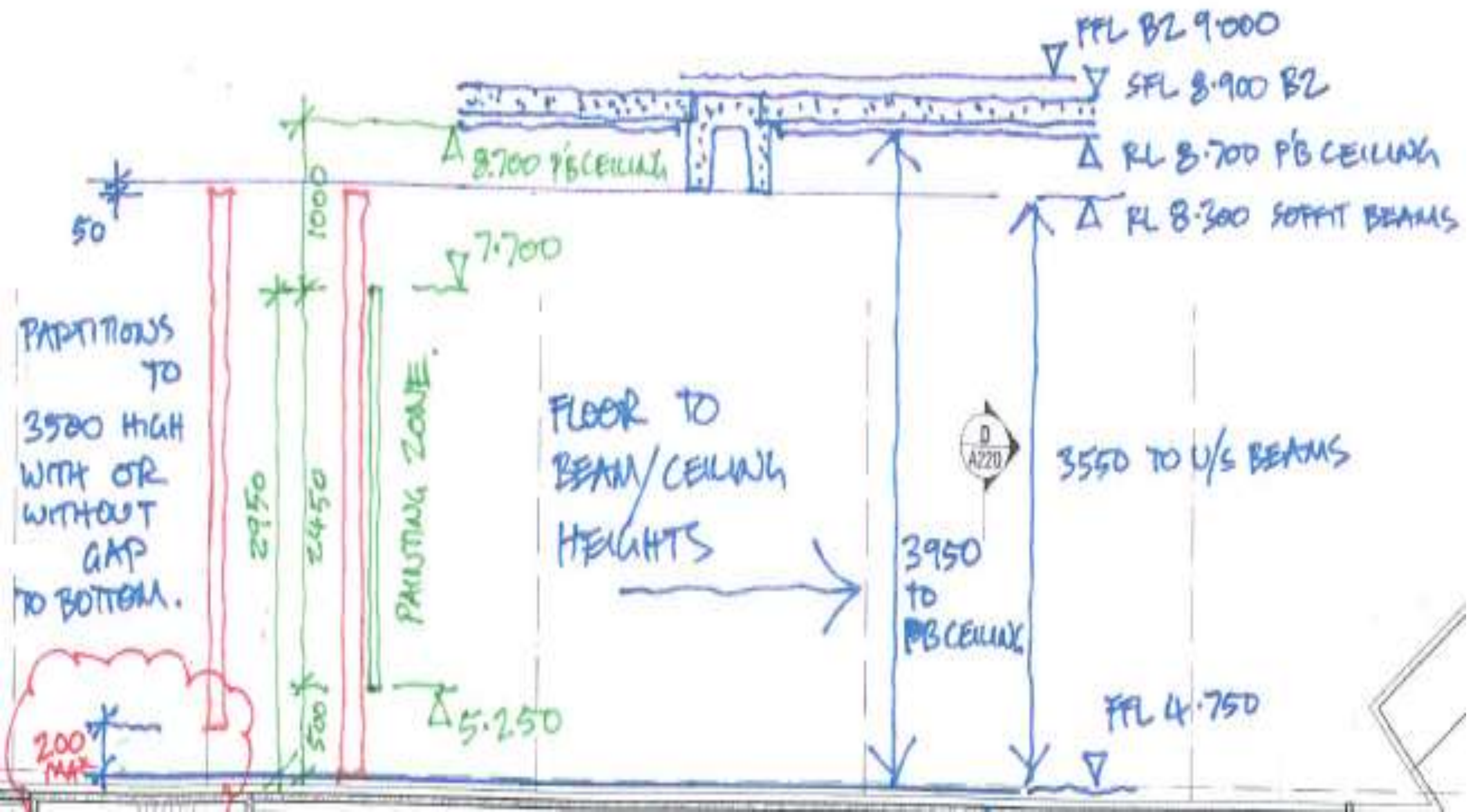
The temperature profile at the bottom of the painting zone

Seminar

CHAM



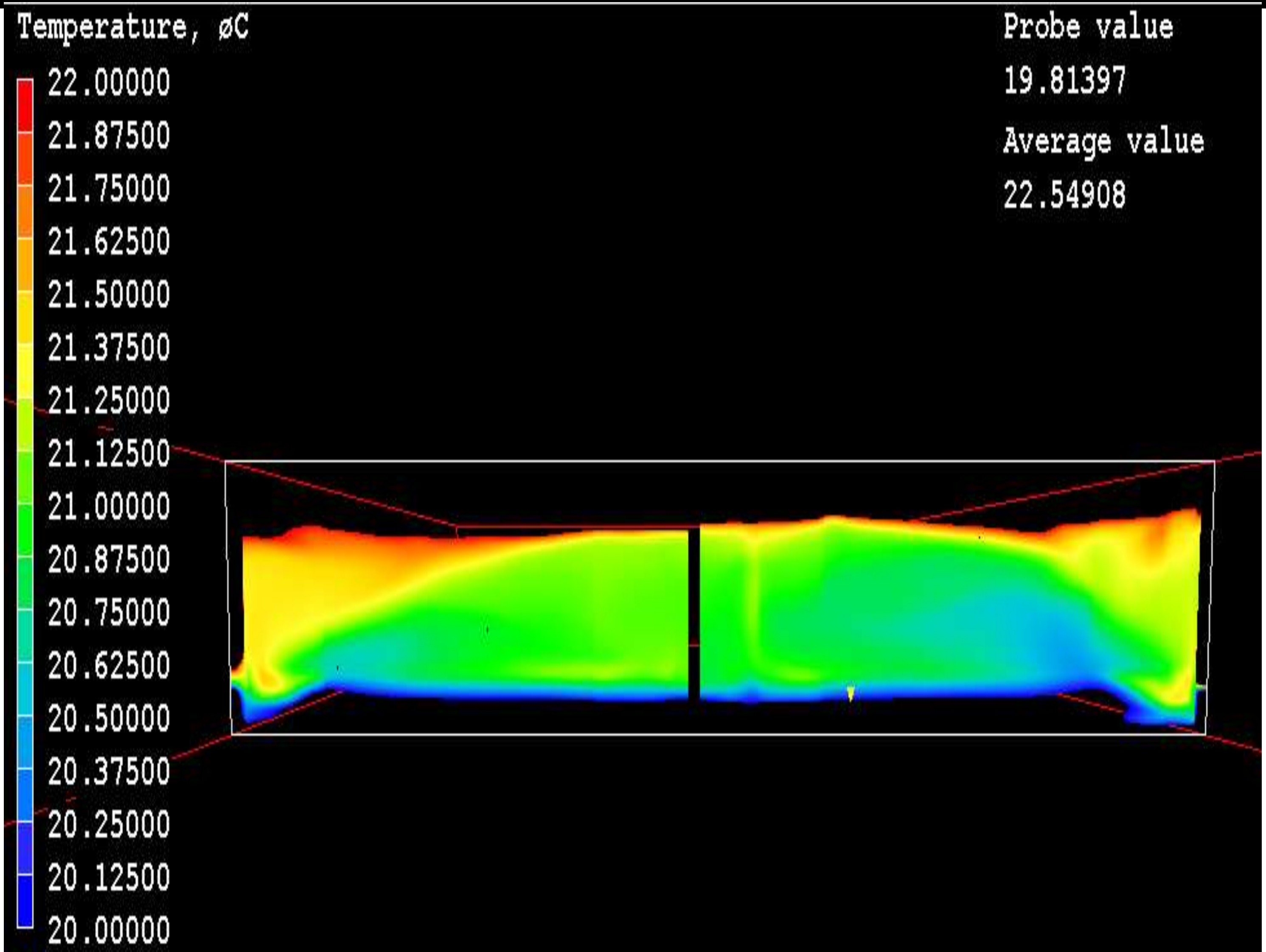
Section drawing



The temperature profile across the painting zone

Seminar

CHAM



Results

	Average Temperature at bottom of painting zone	Average Temperature at top of painting zone	Average Temperature across painting zone	Maximum Velocity 100mm above the floor
	° C	° C	° C	m/s
Option A-1	20.02	21.62	1.6	0.34
Option A-2	19.96	21.59	1.63	0.34
Option A-1-1	20.04	21.77	1.73	0.35

City Link Burnley Stack – Melbourne, Victoria

Seminar

CHAM

The problem

Seminar

CHAM

- One part of the City Link project involved the construction of two three-lane underground tunnels, one 3.4km long and the other 1.6km long, located adjacent to the Melbourne CBD.
- The tunnels incorporate vent stacks that cater for the dispersion of the vehicular exhaust emissions from within the tunnels.
- As part of the Environmental Protection Authority (EPA) policy, the emissions from the vent stacks need to be measured and monitored to ensure that pollutant concentrations are below statutory levels, due to the close proximity of the vent stacks to commercial and residential dwellings.

The problem

CHAM

During the commissioning stages of the tunnel construction, it was discovered that the discharge flow within the exhaust stack where the sampling probes were positioned (midway up the vertical shafts) was non-uniform and unsteady (fluctuating). As a result the sampling probes were unable to be satisfactorily calibrated.

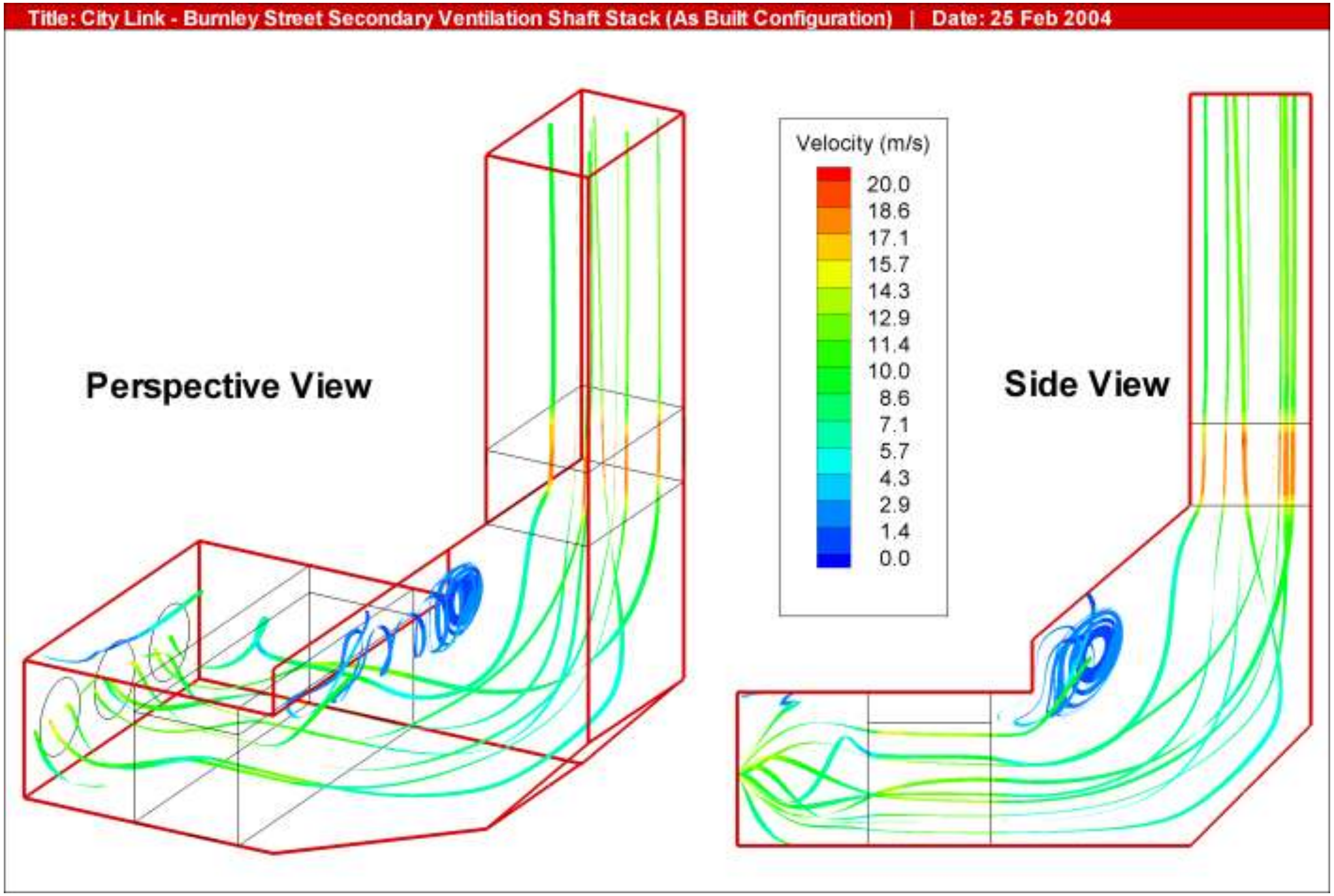
In order to avoid any delay in the opening of the tunnel, along with associated stringent financial penalties to the construction consortium, a solution to this problem had to be found within a very short time scale.

To assist in this process a CFD study was undertaken to establish the flow characteristics through the stacks and to determine a satisfactory solution.

The CFD predictions

Seminar

CHAM



Solutions

The CFD analysis indicated that for the existing as-built configuration the presence of significant flow recirculation within the lower chamber in-between the two banks of attenuators.

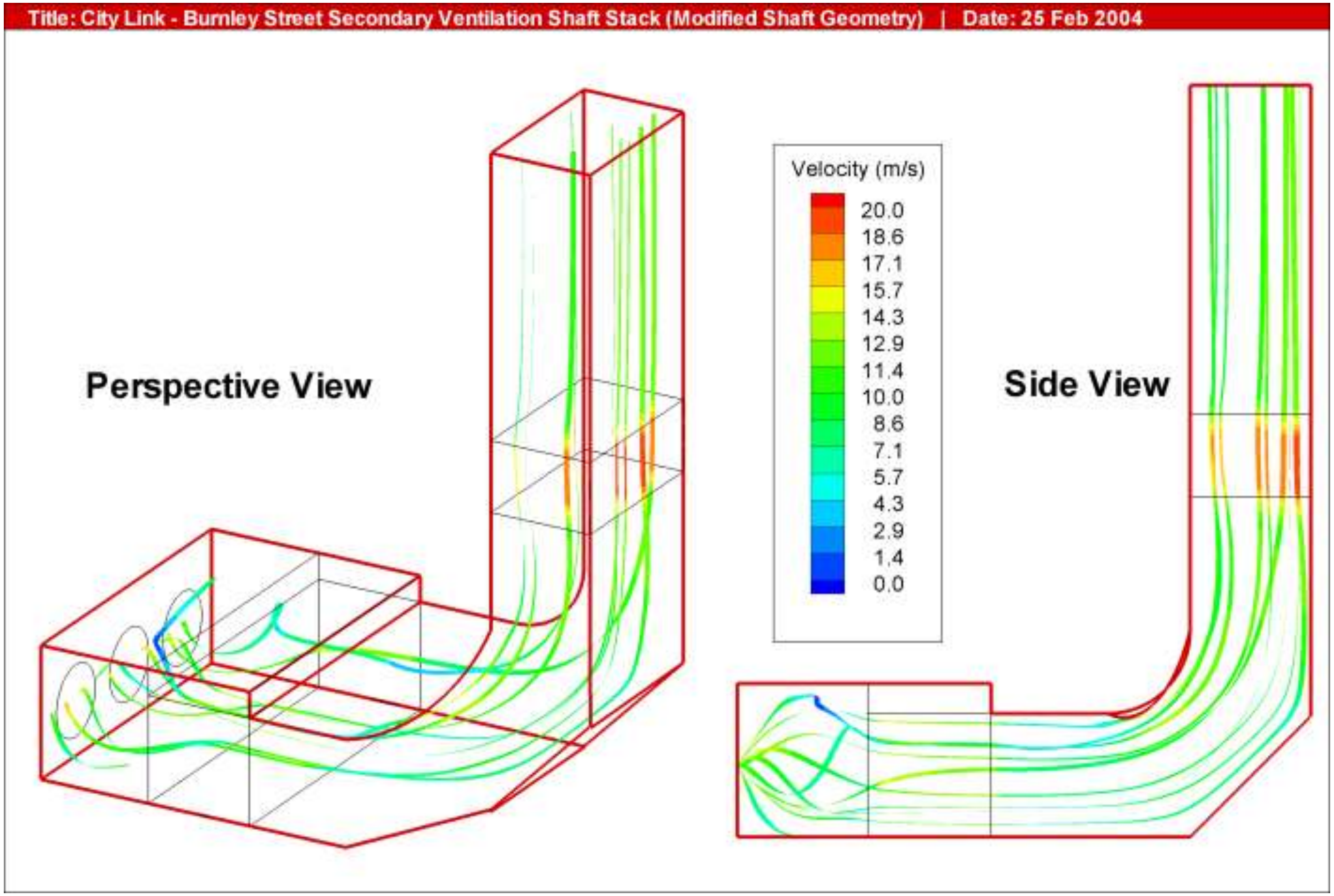
This is caused by the flow expansion into the lower chamber from the first bank of attenuators and extraction fans.

Based upon the results from the CFD analysis, internal fairings and turning vanes were recommended to be installed at the entrance base of the stack.

This was found in practise to eliminate the problem such that the discharge emissions could be satisfactorily monitored to the required EPA regulatory practice.

The improved flow pattern

Seminar



CHAM

Sports Ground Development
Telstra Dome – Melbourne

Seminar

CHAM

Data and graphics courtesy of

ConnellWagner aurecon

- **Located within Melbourne Docklands urban re-development area.**
- **Designed for AFL, soccer, rugby, cricket and concerts.**
- **Seating capacity of 52,000 with a movable tier of 12,500 seats.**
- **167m by 132m retractable roof**

Sports Ground Development
Telstra Dome – Melbourne

Seminar

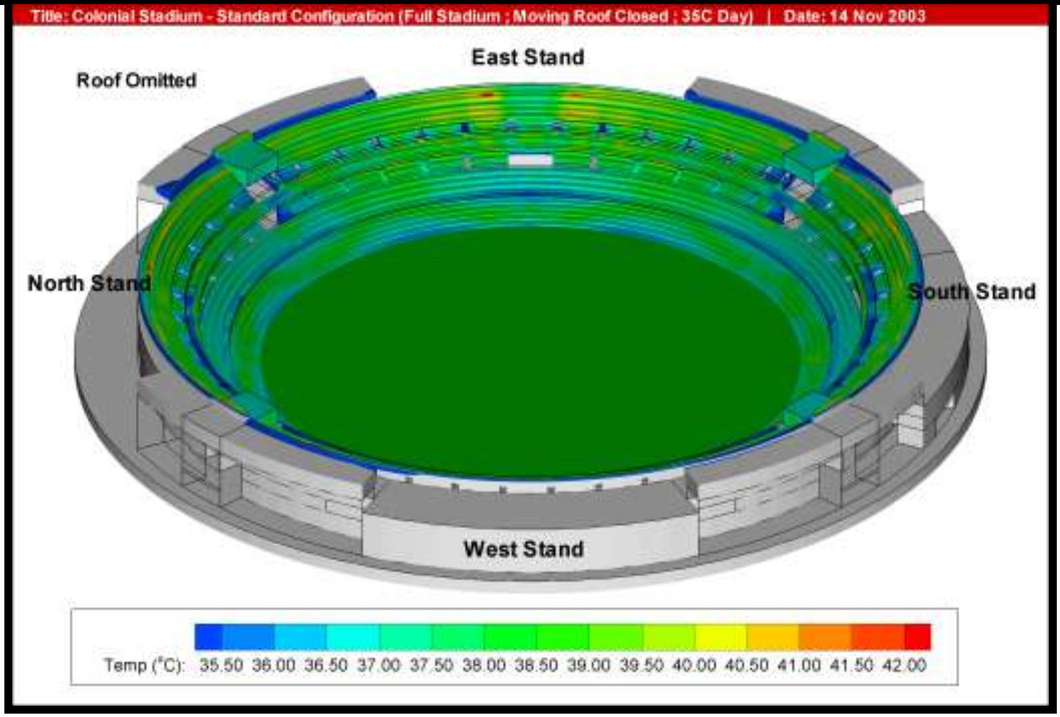
CHAM

- **When the roof is closed the stadium becomes a fully enclosed all weather 'indoor' facility.**
- **Occupancy comfort and safety needed consideration.**
- **Similar stadia often have mechanical ventilation.**
- **Concerns re large capital cost and ongoing energy consumption led to a study to justify an effective passive (natural) ventilation solution.**

Sports Ground Development
Telstra Dome – Melbourne

Seminar

CHAM

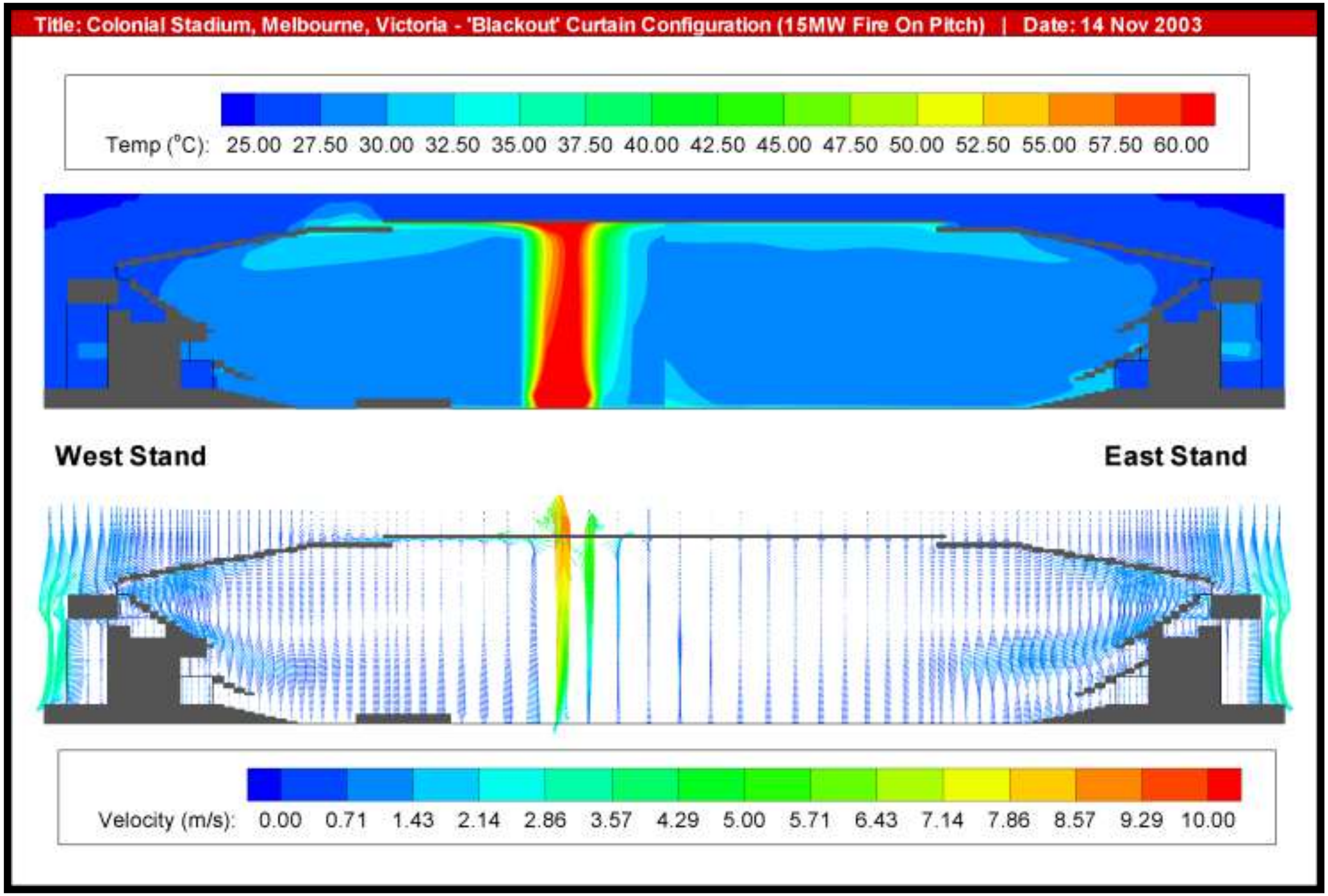


- This slide shows the temperature distribution for a perceived 'worst case' condition scenario of a hot day (35 ^{o}C) with no wind and the stadium roof closed.
- The observed maximum temperature rises in the seating areas are around 4 - 5^{o}C above the ambient temperature.

Sports Ground Development
Telstra Dome – Melbourne

Seminar

CHAM



- Location and spacing of roof vents was optimised.
- Architectural design of roof vents and façade openings to promote air movement which enhances occupant comfort for a wide range of environmental conditions.
- Temperature rises around bowl area not deemed excessive (within 5^{o}C).
- Life safety tenability criterion satisfied.

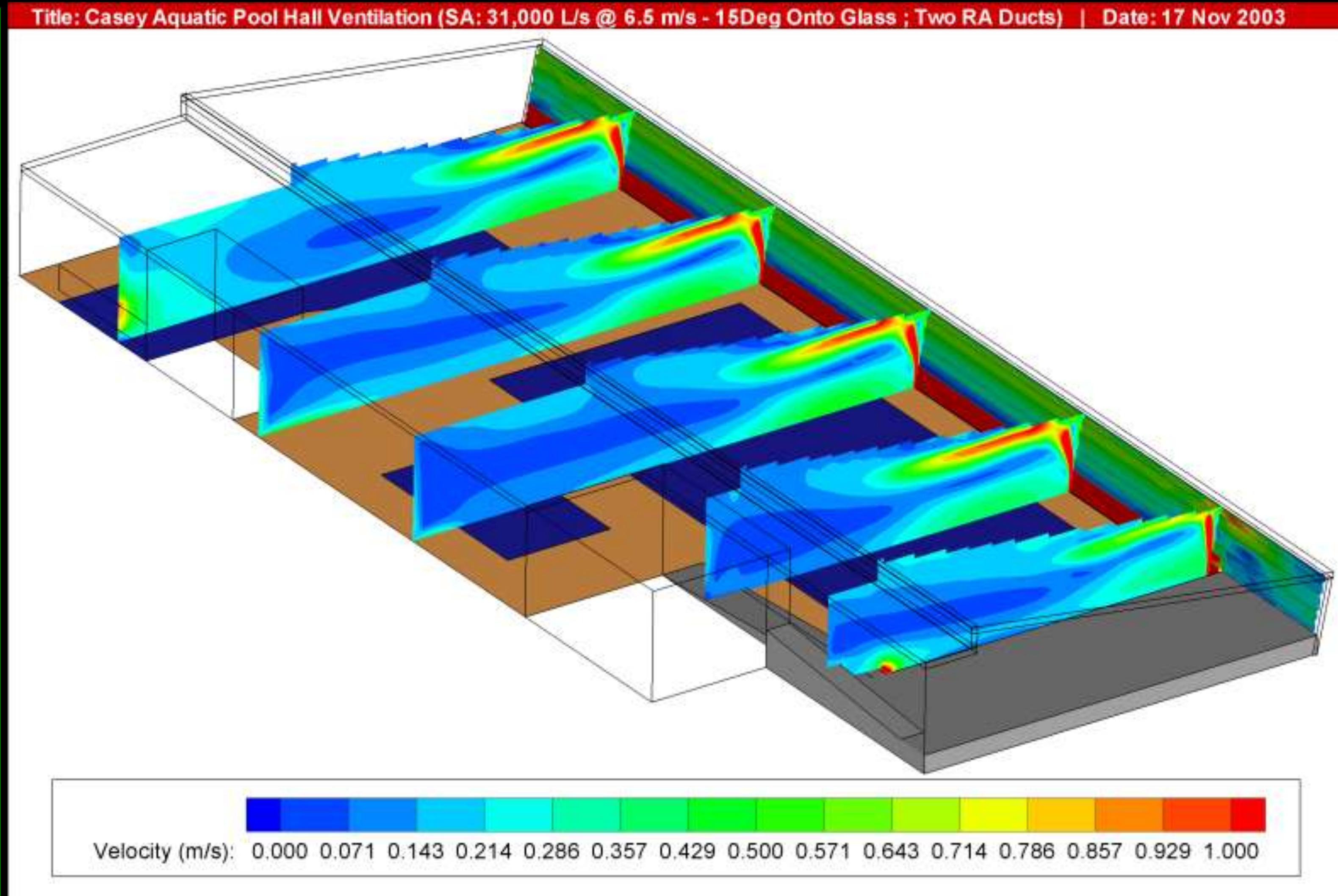
Casey Aquatic Centre - Melbourne, Australia

- Ventilation within an indoor pool needs to ensure sufficient "fresh" air requirements
- Necessary for removal of water vapour and chemical vapours
- Different mechanical ventilation systems configurations were analysed with CFD
- Establish air movement distribution and show undesirable stagnant flow regions

Casey Aquatic - Design Analysis
Mechanical ventilation-"push-pull" system

Seminar

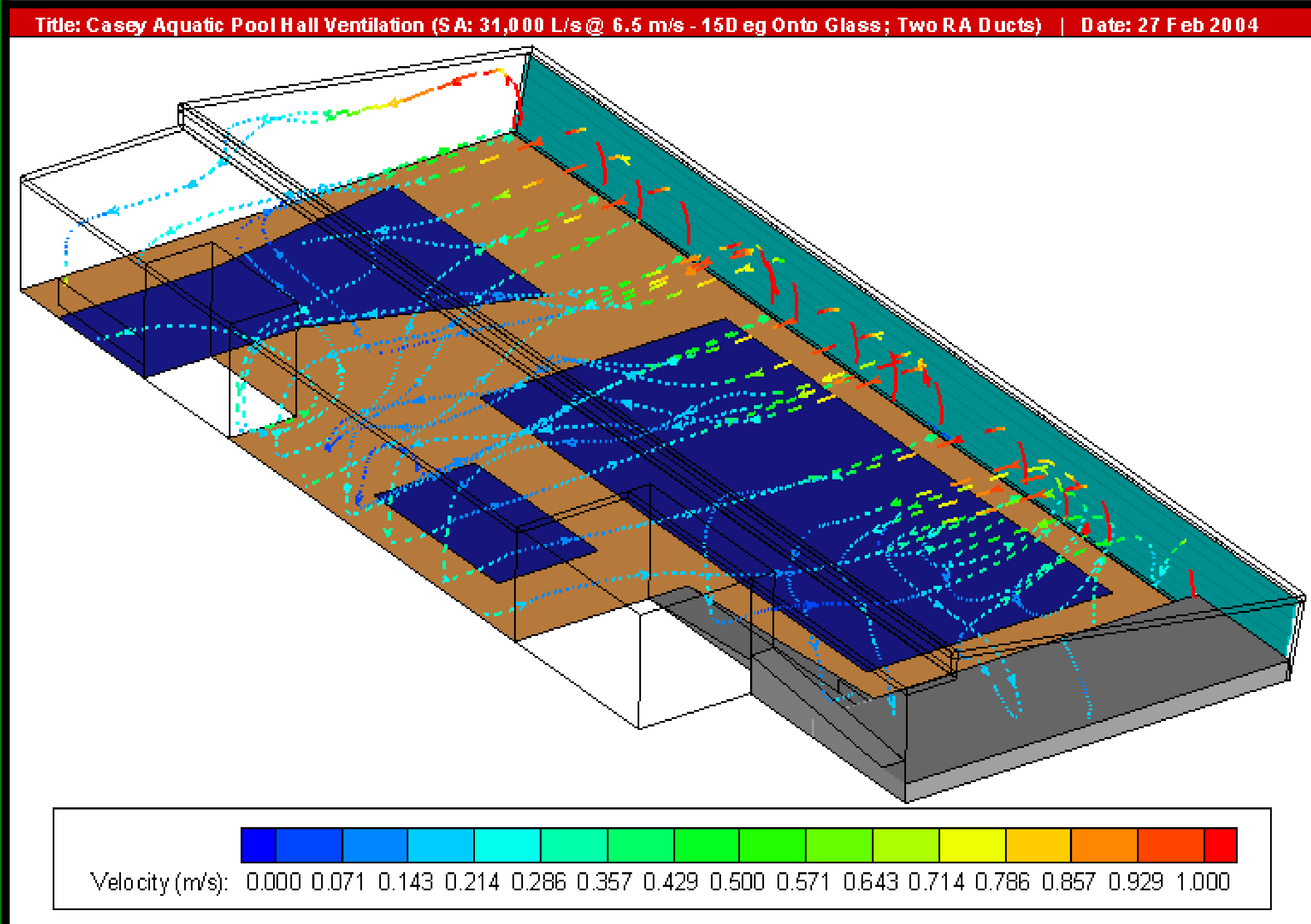
CHAM



Casey Aquatic - Design Analysis
Mechanical ventilation-"push-pull" system

Seminar

CHAM



MCG Redevelopment - Melbourne, Australia

Seminar

CHAM

MCG Redevelopment - Design Scope

Seminar

CHAM

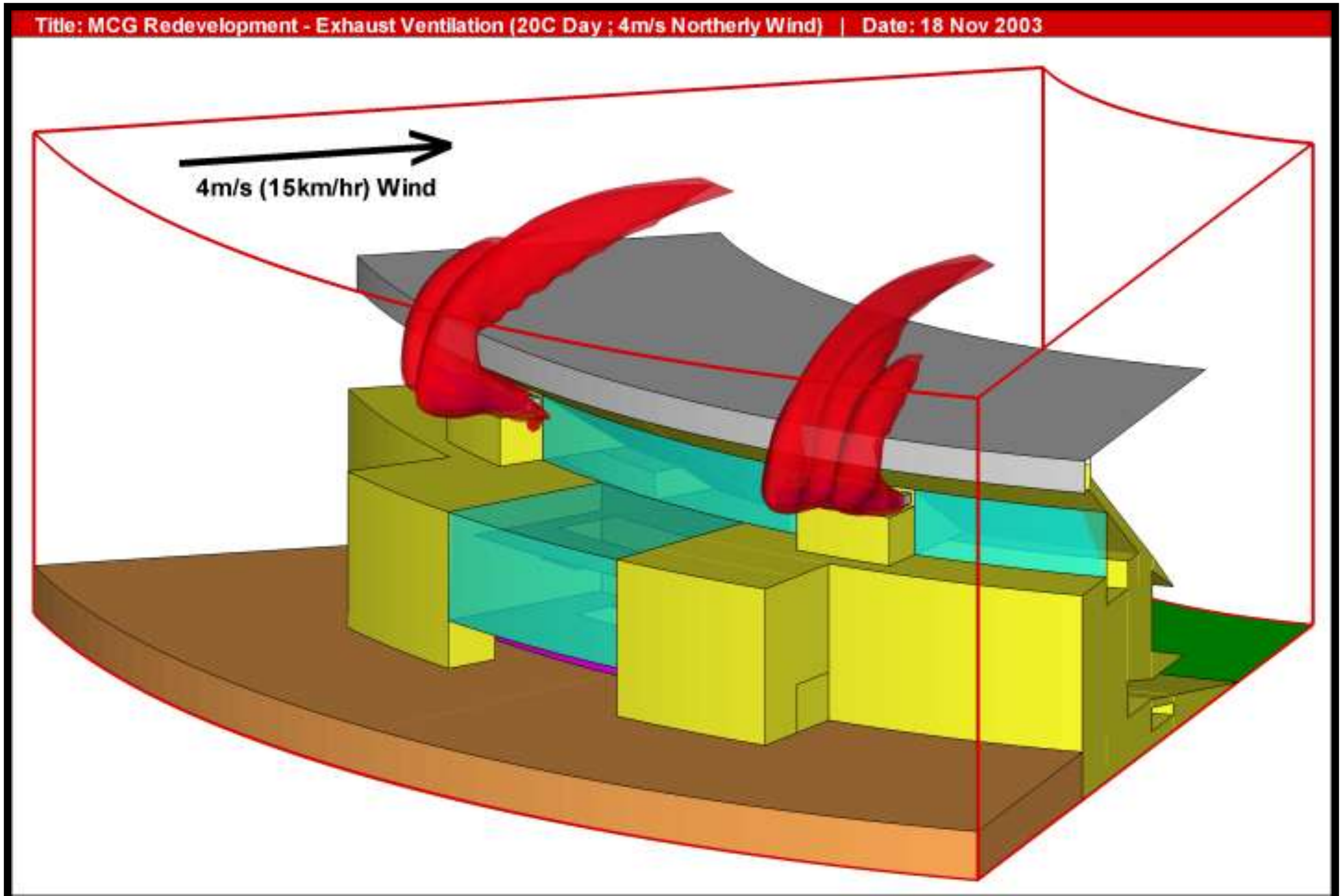
- Several CFD modelling studies done for:-
- Northern side of stadium being redeveloped for utilisation at the Commonwealth Games
  - Ventilation (thermal comfort)
  - Plant room exhaust dispersion
  - Pitch ventilation
  - Fire life safety

MCG Redevelopment - Design Analysis

Seminar

CHAM

Plant Room Exhaust : 4m/s Northerly Wind

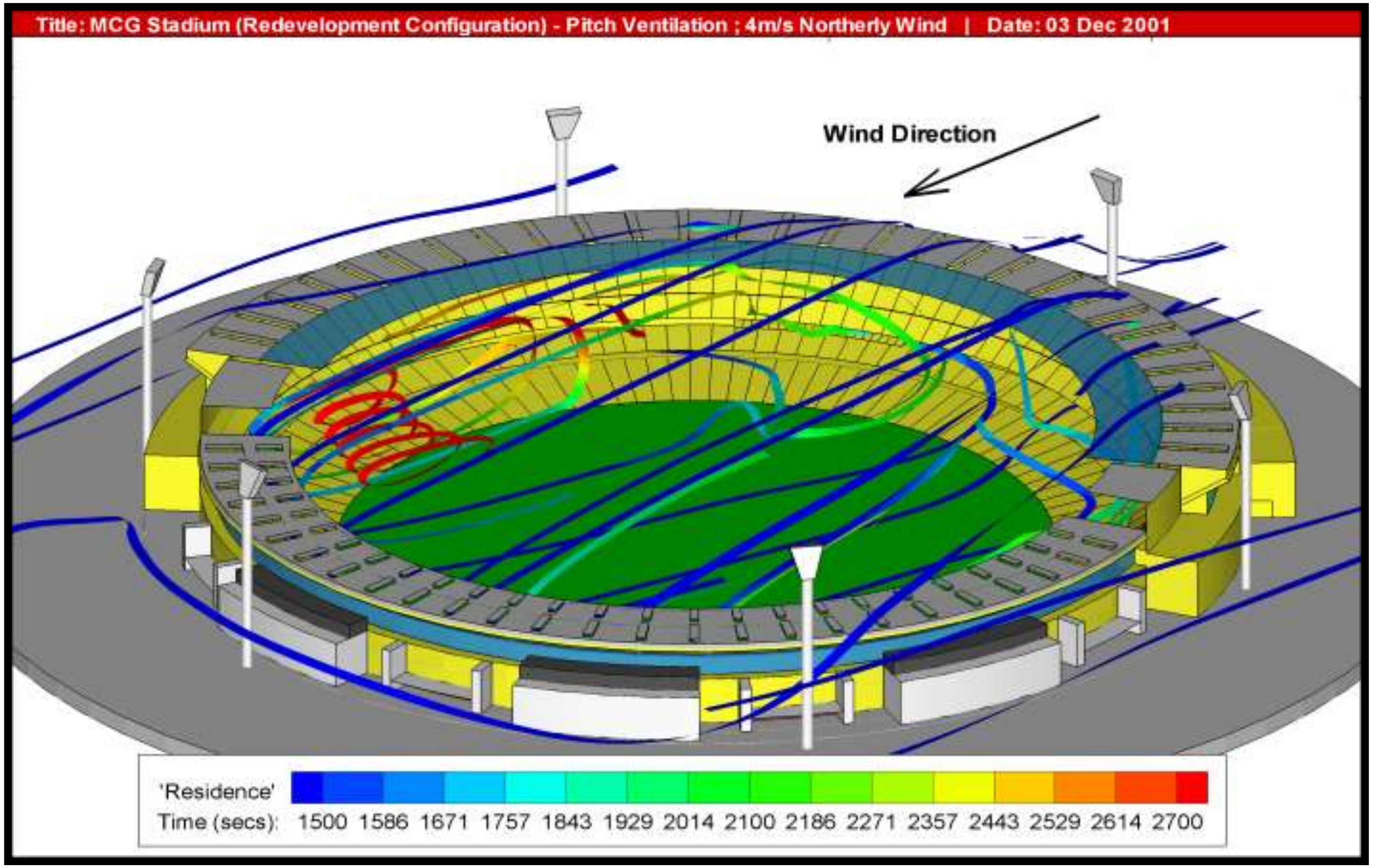


MCG Redevelopment - Design Analysis

Seminar

CHAM

Pitch Ventilation : 4m/s Northerly Wind



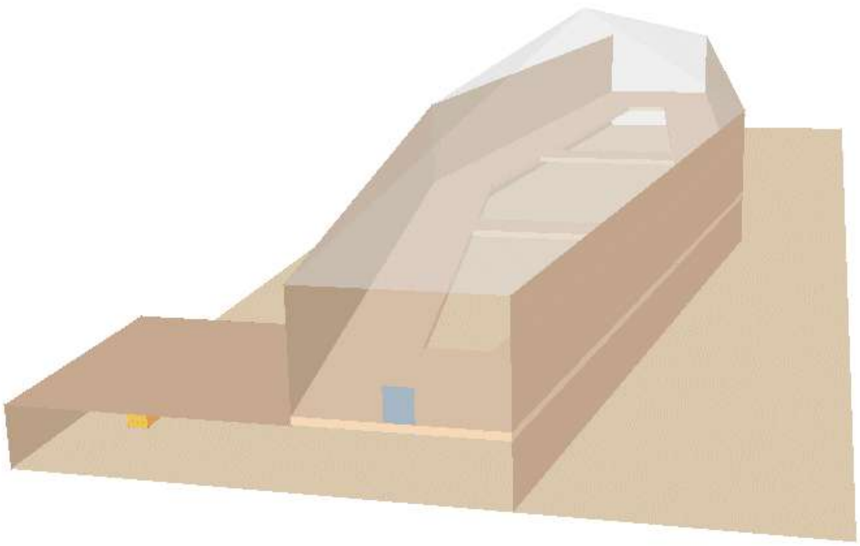
Conclusions

- Project case studies have shown the use of CFD as an important part of the engineering design process of sports stadia/arenas
- Issues relating to ventilation, occupancy comfort and life safety can be addressed
- Facilitates 'sustainable design technology' solutions by avoiding/limiting mechanical ventilation energy usage as far as possible
- Can be promoted as world's 'best practice' and used as a benchmark for stadia design

Madrid Xanadu Shopping Mall Fire Study

Seminar

CHAM



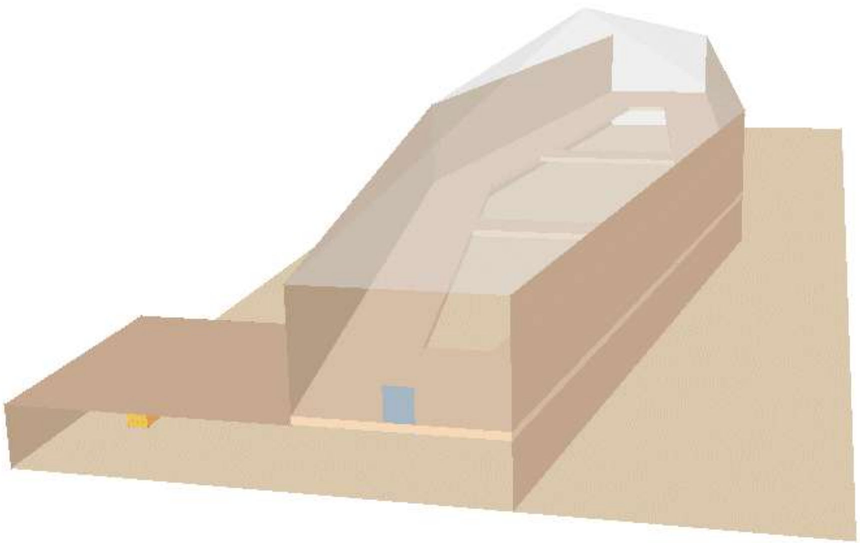
During the design of the Xanadu Shopping Mall near Madrid, Spain, concerns were expressed about the safety of the food hall in the event of a fire.

Simulations to address this issue were carried out on behalf of LWF - Fire Engineering and Fire Risk Management Consultants.

Madrid Xanadu Shopping Mall Fire Study

Seminar

CHAM



The design of the food hall is conventional, with two levels; openings in the first floor add to the feeling of 'open space' for shoppers.

However, the building is longer than previous similar structures: the central space is 139m long, 33m wide and 24m high.

These dimensions meant that the roof space provided a smoke reservoir in excess of the conventional guidelines for such buildings.

Madrid Xanadu Shopping Mall Fire Study

Seminar

CHAM

At one end of the hall there is a small door on the upper floor, while the other end links to the rest of the shopping mall via a large open walkway on each level.

The major concern was that hot air and smoke from a fire may prevent escape from the upper level of the food hall into the rest of the complex.

A further complication was added by the legislative requirement that smoke control measures for new buildings should be achieved by natural, rather than mechanical, methods.

The proposed design solution was the introduction of a large number of vents near the top of the side walls, just below the base of the domed roof space.

The simulations were intended to show whether the original fears about smoke behaviour were justified and, if so, whether the additional vents would provide an acceptable improvement in safety.

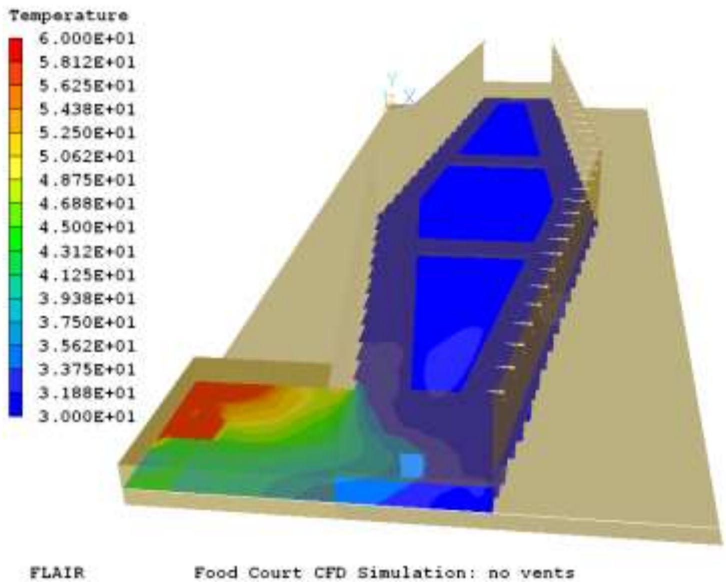
The simulated scenario was for a fire in one of the end units on the lower level of the hall, furthest from the escape route (as shown in the figure).

The size of the fire was 2.5MW, with only the natural ventilation available through the ends of the hall (plus the vents, when included) to dissipate the heat.

Madrid Xanadu Shopping Mall Fire Study

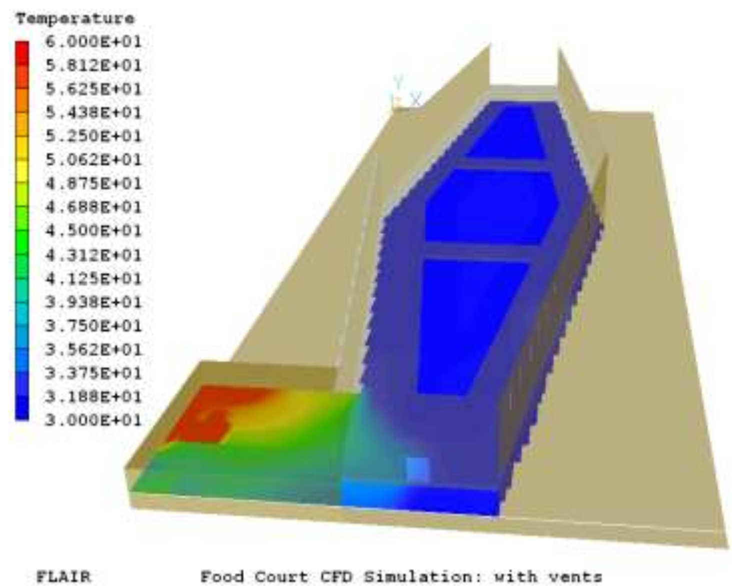
Seminar

CHAM



- Temperature contours at head height on lower level - no vents

- Temperature contours at head height on lower level - with vents

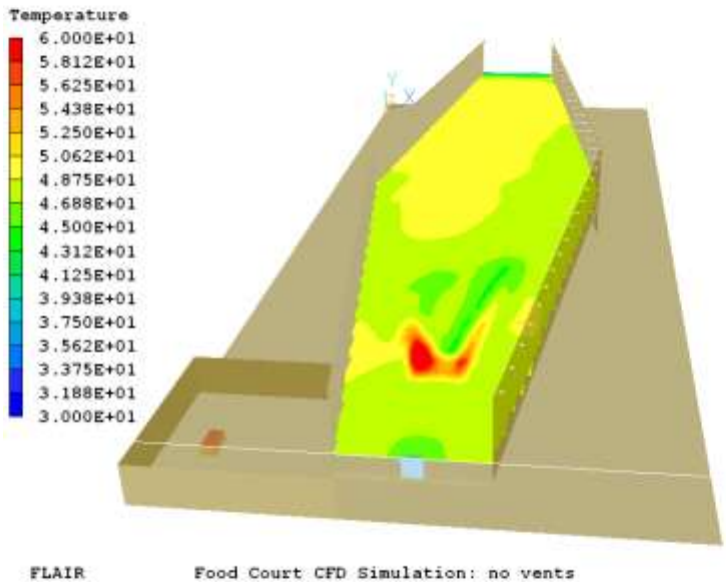


Madrid Xanadu Shopping Mall Fire Study

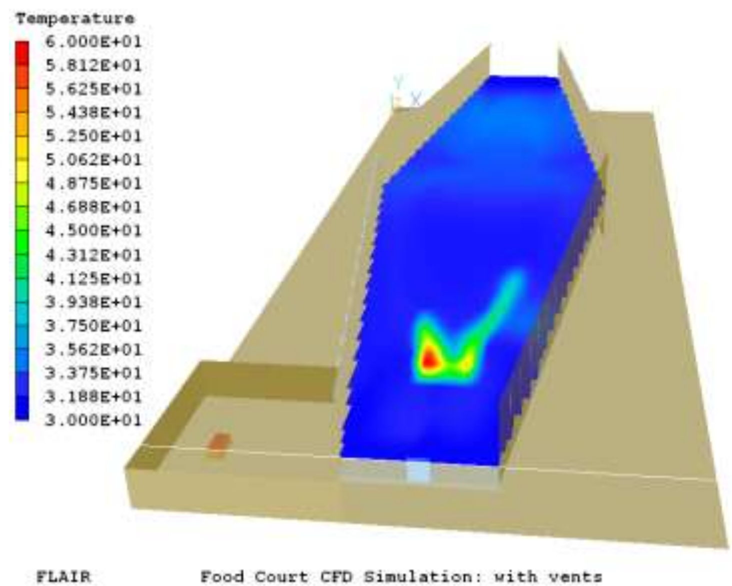
Seminar

CHAM

- Temperature contours at head height on upper level - with vents



- Temperature contours at head height on upper level - no vents



There is not much difference in the temperatures on the lower floor.

It is clear that the temperature is dangerously high on the upper floor when there are no vents, and that the vents reduce this to a level which is little higher than the ambient temperature (30ºC).

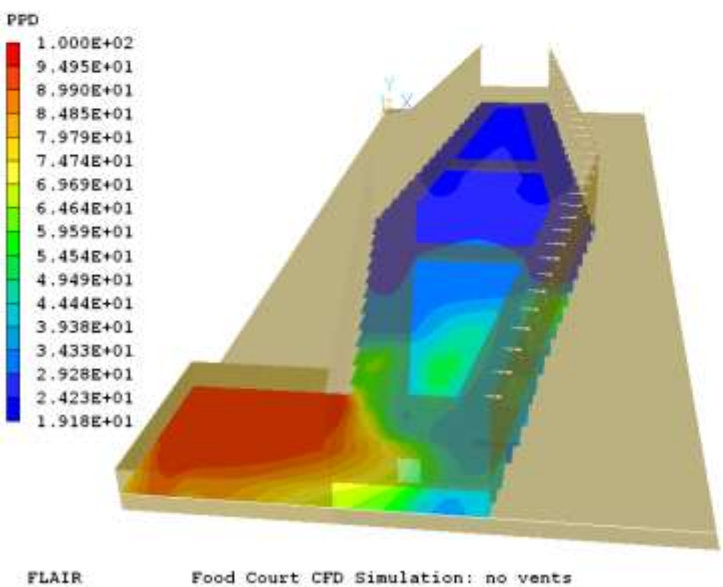
The next pictures show the PPD (Predicted Percentage Dissatisfied) contours.

Madrid Xanadu Shopping Mall Fire Study

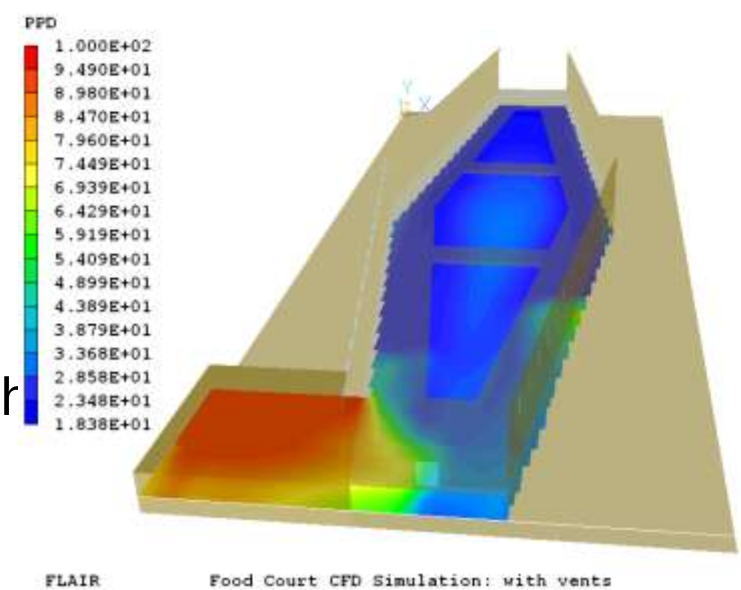
Seminar

CHAM

- PPD contours at head height on lower level - with vents



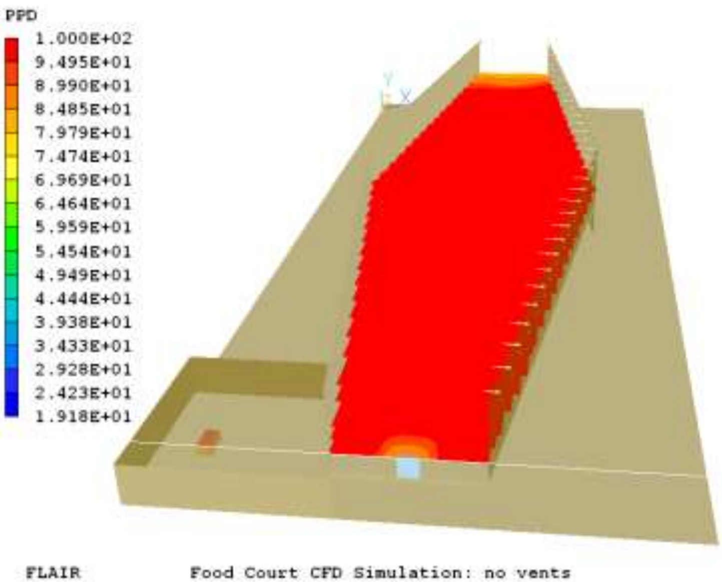
- PPD contours at head height on lower level - no vents



Madrid Xanadu Shopping Mall Fire Study

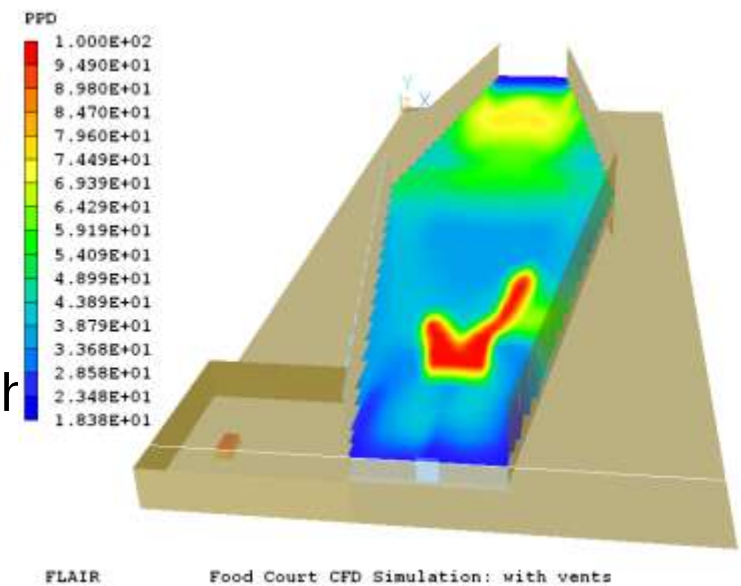
Seminar

CHAM



- PPD contours at head heigh on upper level - no vents

- PPD contours at head height on upper level - with vents



Again, not too much difference on the lower floor, although a higher percentage of the floor area is uncomfortable.

A huge difference on the upper floor, where the vents reduce the PPD from 100% to a much lower level over most of the floor area.

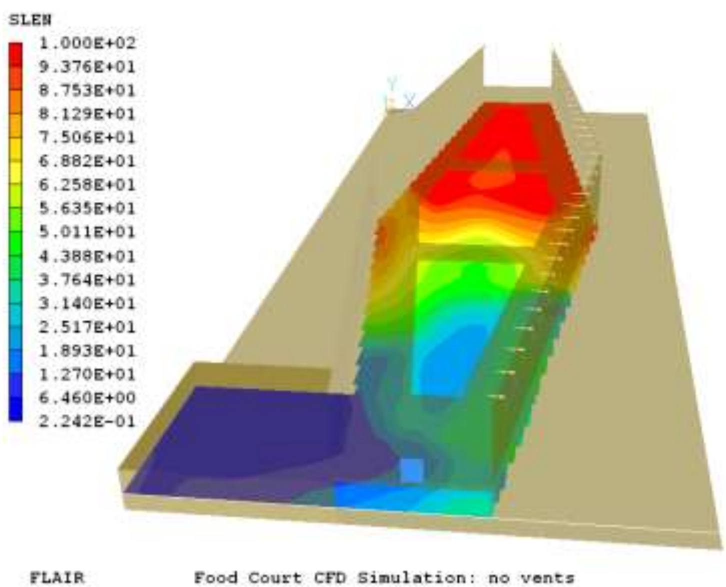
The next pictures show the visibility contours.

Madrid Xanadu Shopping Mall Fire Study

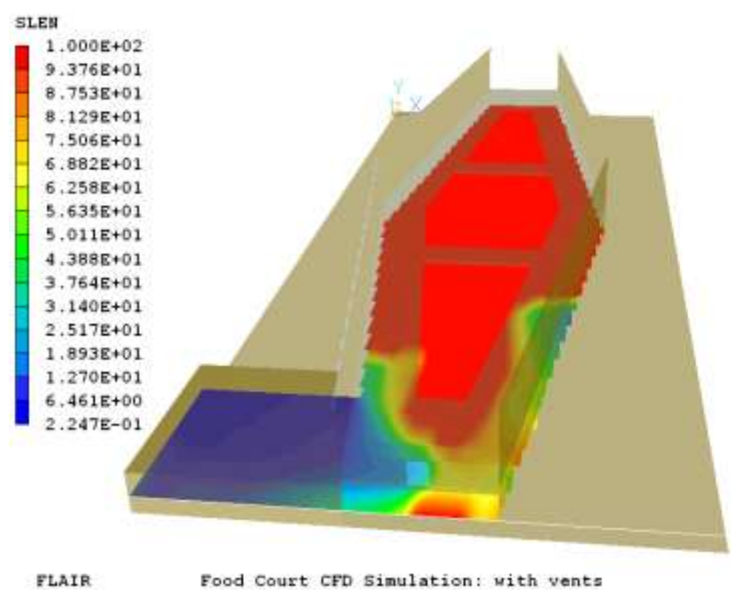
Seminar

CHAM

- Visibility contours at head height on lower level - with vents



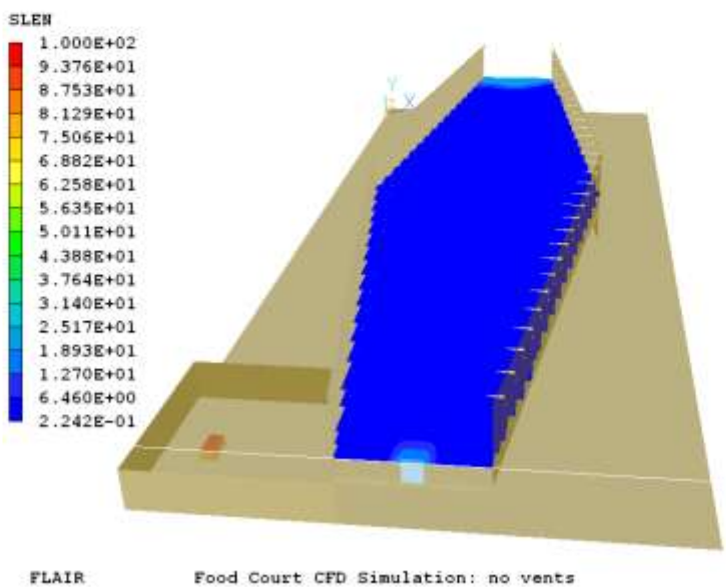
- Visibility contours at head height on lower level - no vents



Madrid Xanadu Shopping Mall Fire Study

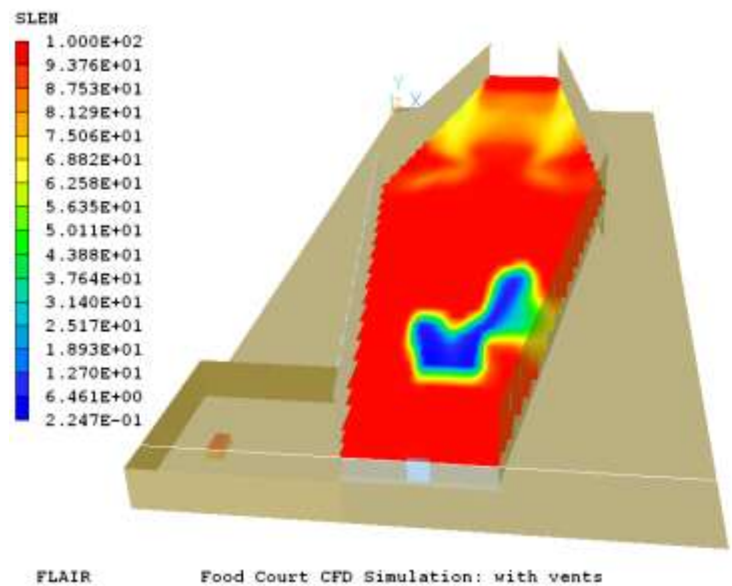
Seminar

CHAM



- Visibility contours at head height on upper level - with vents

- Visibility contours at head height on upper level - no vents



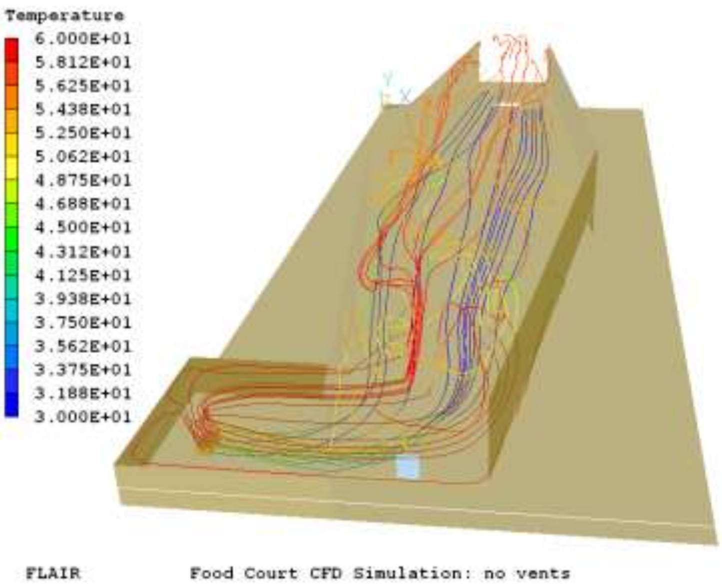
On the lower level, visibility away from the fire zone is not too bad in either case.

On the upper level, visibility is very poor in the case with no vents.

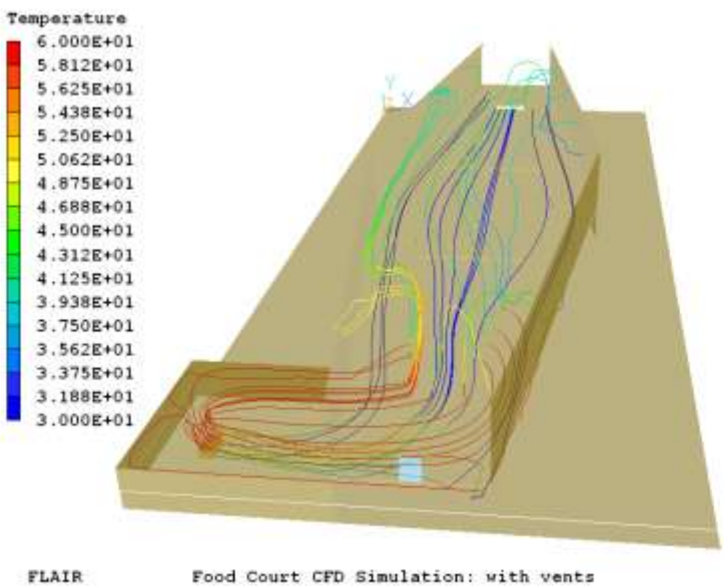
Madrid Xanadu Shopping Mall Fire Study

Seminar

CHAM



- Streamlines emanating from the fire - with vents

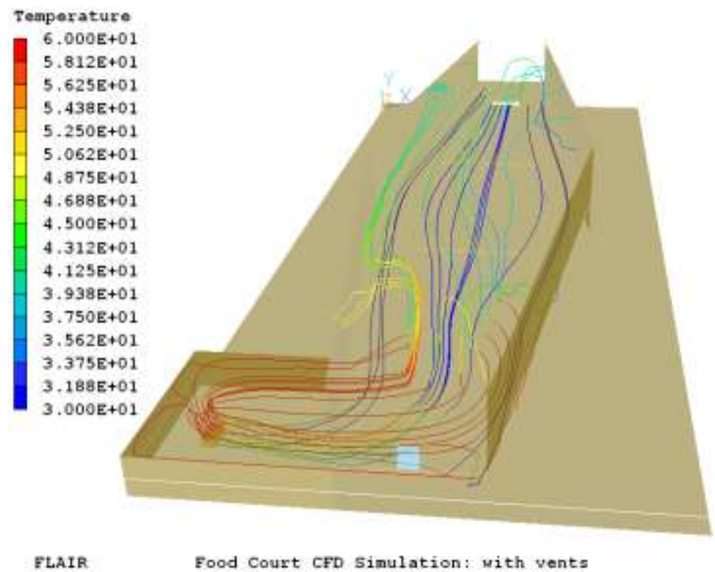


- Streamlines emanating from the fire - no vents

Madrid Xanadu Shopping Mall Fire Study

CHAM

The reason for the difference in the temperature contours is clear. Without the vents the hot and smoky air fills the domed roof and can only escape through the walkway.



The vents enable the hot air to escape easily: in fact, the number, or size, could easily be reduced without compromising the safety of the building.

Note the blue streamlines, showing the path of the air before it is entrained into the fire: it is drawn in along the full length of the lower level of the hall.

Madrid Xanadu Shopping Mall Fire Study

The PHOENICS simulations enabled a good understanding of the air flow in the food hall to be obtained, under the assumed fire conditions.

The effectiveness of the high-level vents could be demonstrated, enabling the modified design to be validated.

The whole package of fire design measures, of which the smoke control was a part, resulted in an estimated saving of about 250,000 Euros - and a solution more suited to the environment.

Madrid Xanadu Shopping Mall Fire Study –Technical Details

The fire was simply specified using a FIRE object as a heat source of 2.5MW, distributed over an arbitrary volume of 1.5m x 3.0m x 1.0m (height), placed inside the shop unit.

The mass-release rate of combustion product was estimated from the assumed heat-release rate and a heat of combustion.

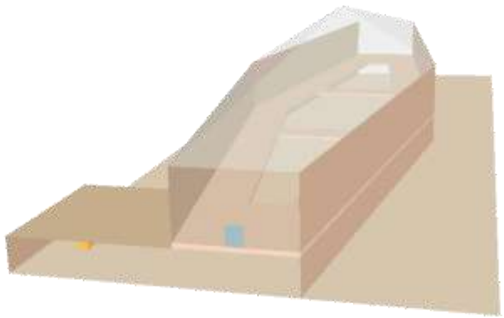
The smoke value for the combustion products was set to 1.0, so that values elsewhere can be used to calculate the smoke density.

The LVEL wall-distance-based model was used for turbulence.

The air was treated as an ideal gas, with buoyancy based on density difference relative to the ambient external temperature (30ºC).

Madrid Xanadu Shopping Mall Fire Study –Technical Details

The shape of the Shopping Mall is reasonably simple, which meant that it could be constructed in a number of different ways using PHOENICS.



The simplest is probably to create the required open space by filling the rest of the solution domain with simple shaped objects: rectangular boxes and wedges.

The dividing floor can be a solid object, with later-defined objects made of air to provide the openings.

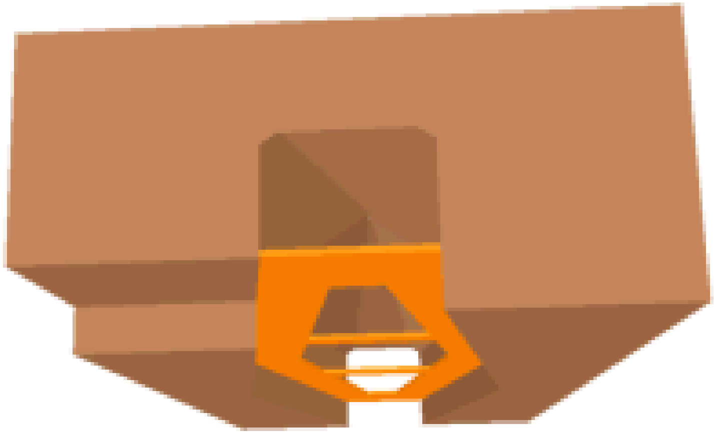
This is a perfectly acceptable way to generate the required geometry, and will produce good results.

Madrid Xanadu Shopping Mall Fire Study –Technical Details

However, the result is somewhat cumbersome, with a large number of objects to be manipulated.

Instead, a slightly more complicated approach was adopted: the non-participating region was constructed using AC3D, a CAD utility provided with PHOENICS.

By this means a single geometry file could be produced, enabling the whole of the geometry to be loaded as one object.



Madrid Xanadu Shopping Mall Fire Study –Technical Details

The visual display of the results is difficult, whichever of the two methods is used, requiring the hiding of various objects which means that the shape of the structure is not clear.

To overcome this, a suite of special 'viewing objects' was constructed, again using AC3D; these were used to provide a less obstructed view of the results in the VR Viewer (post-processor).

CHAM

PHOENICS

Car park Example

Car Park

This work was carried out as a Consultancy model-build in support of ExcelAir BV in Holland.

It concerns the spread of smoke through a complex car park as a result of a car fire on one of the lower floors.

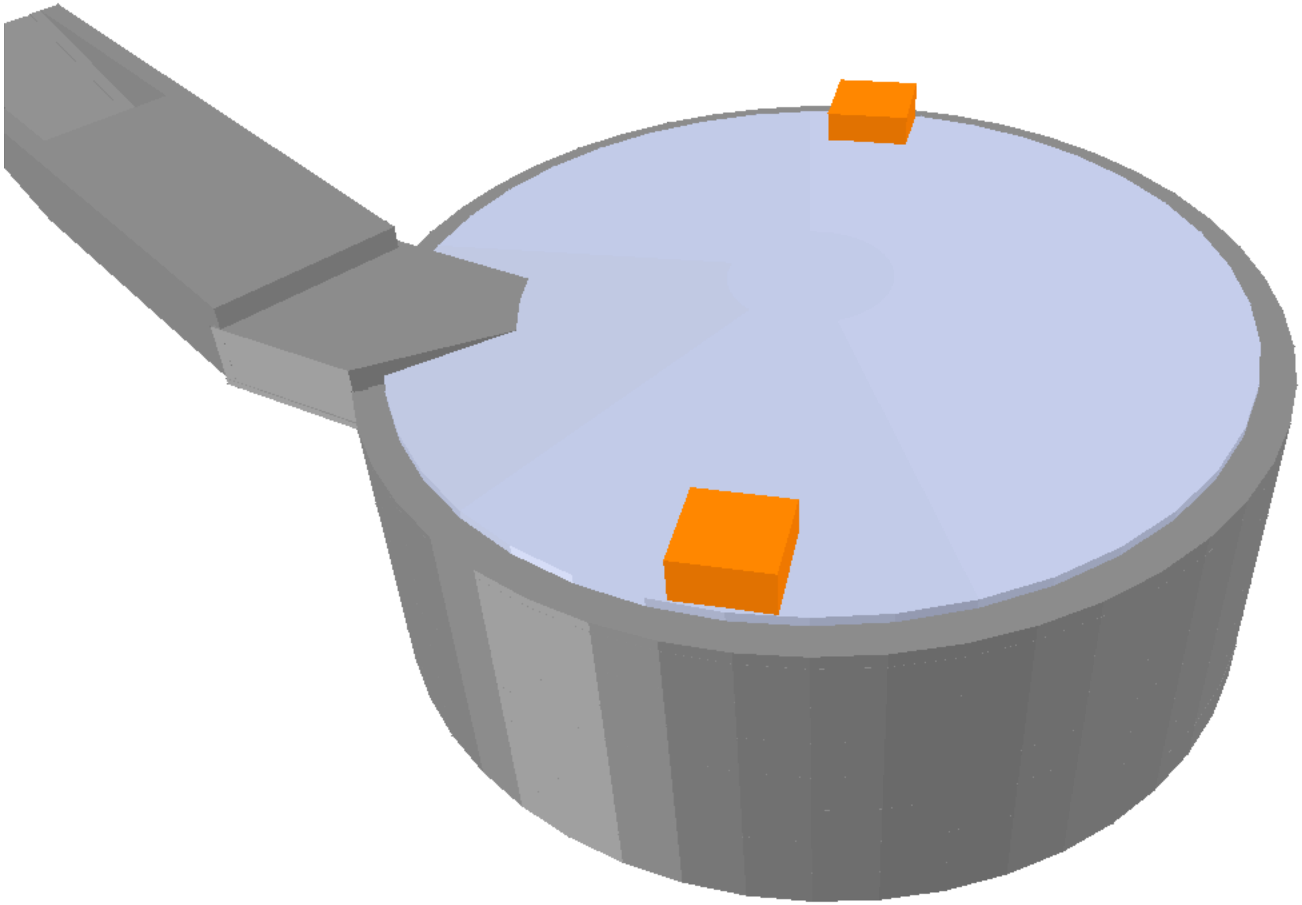
The car park is spiral in form, with an open centre.

The car fire is 8.3 MW. After 180s, sprinklers on the floor where the fire is are activated.

Car Park - Geometry

CHAM

- Roof in place

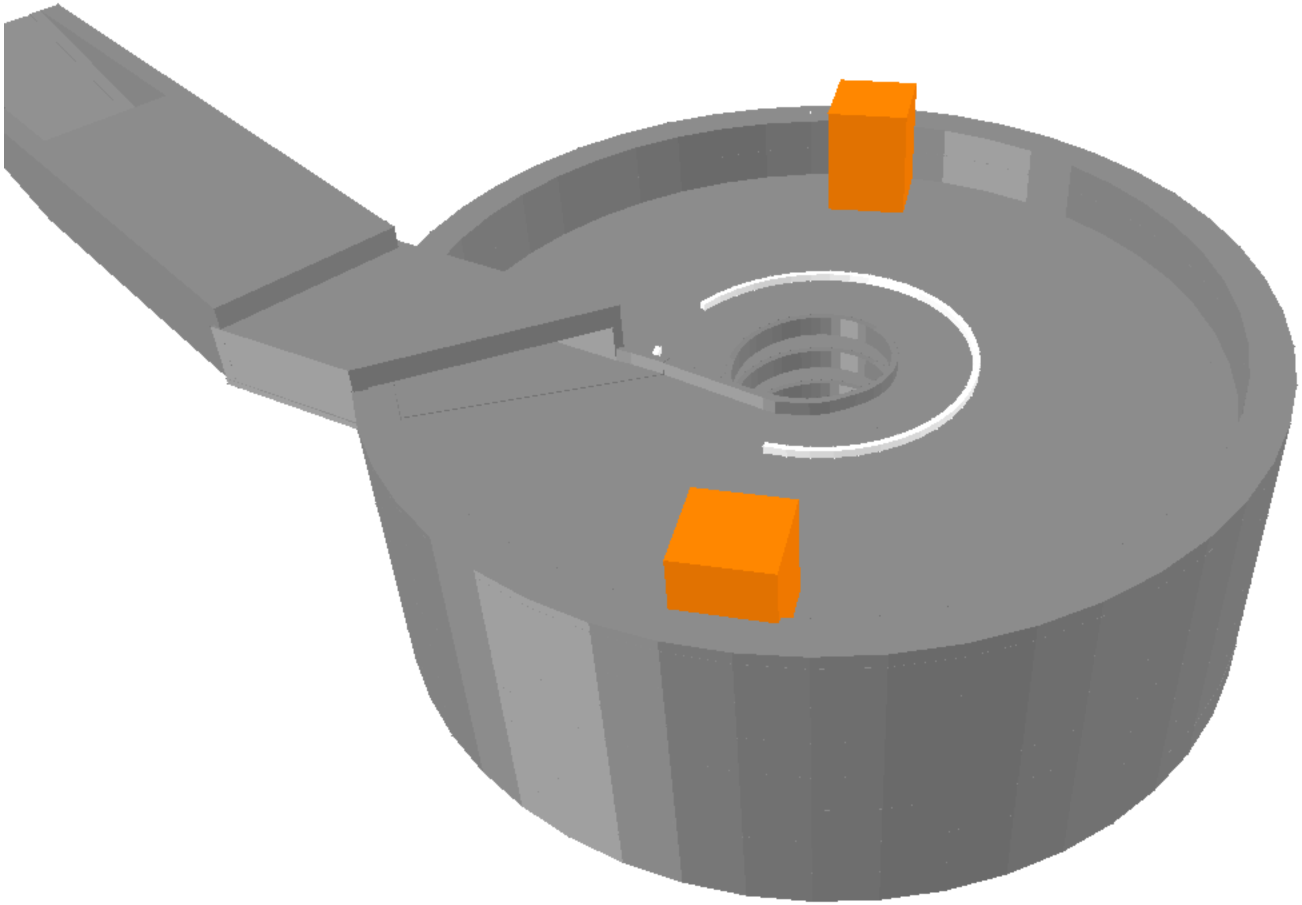


Car Park - Geometry

Seminar

CHAM

- Roof removed

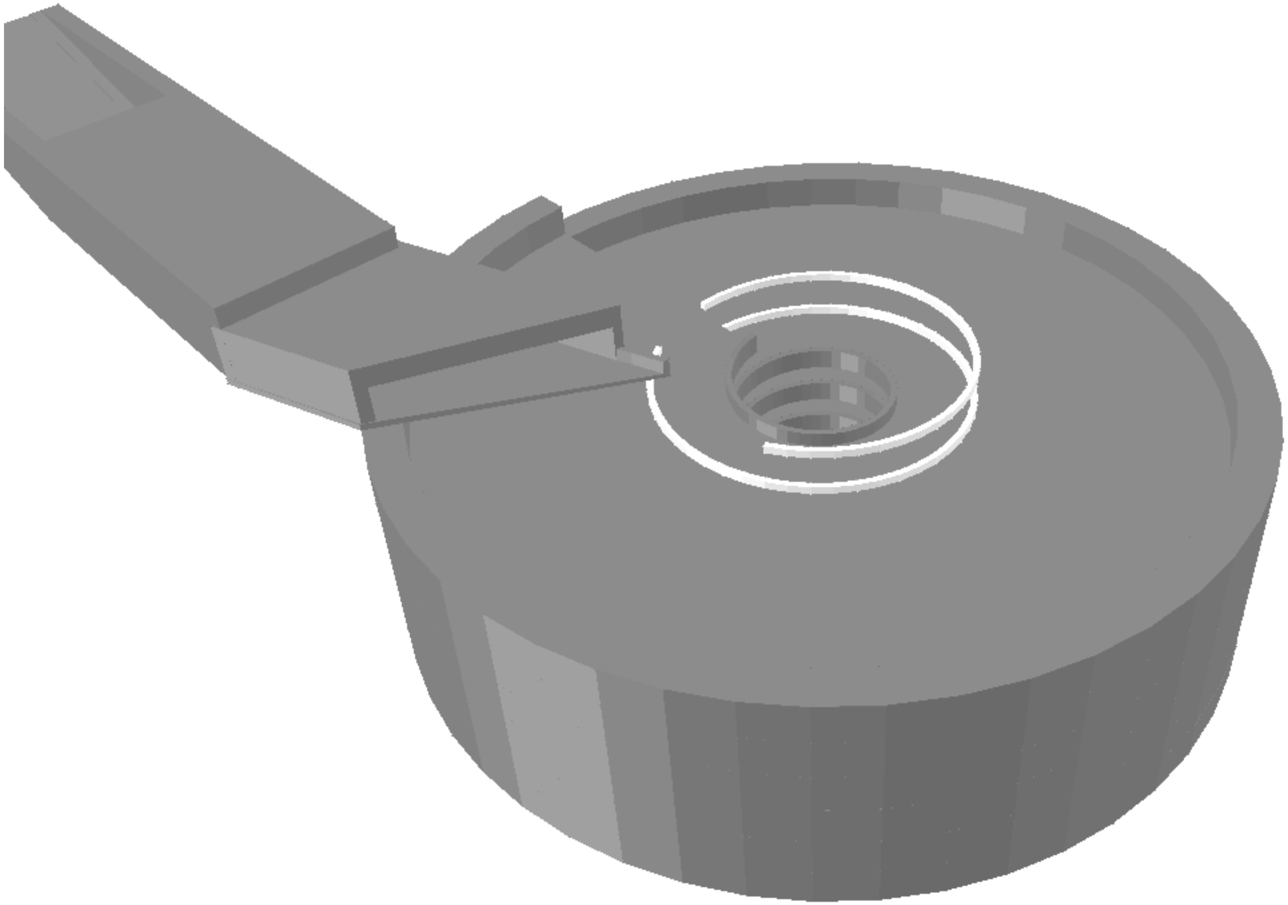


Car Park - Geometry

Seminar

CHAM

- Roof, lift shafts and first floor removed

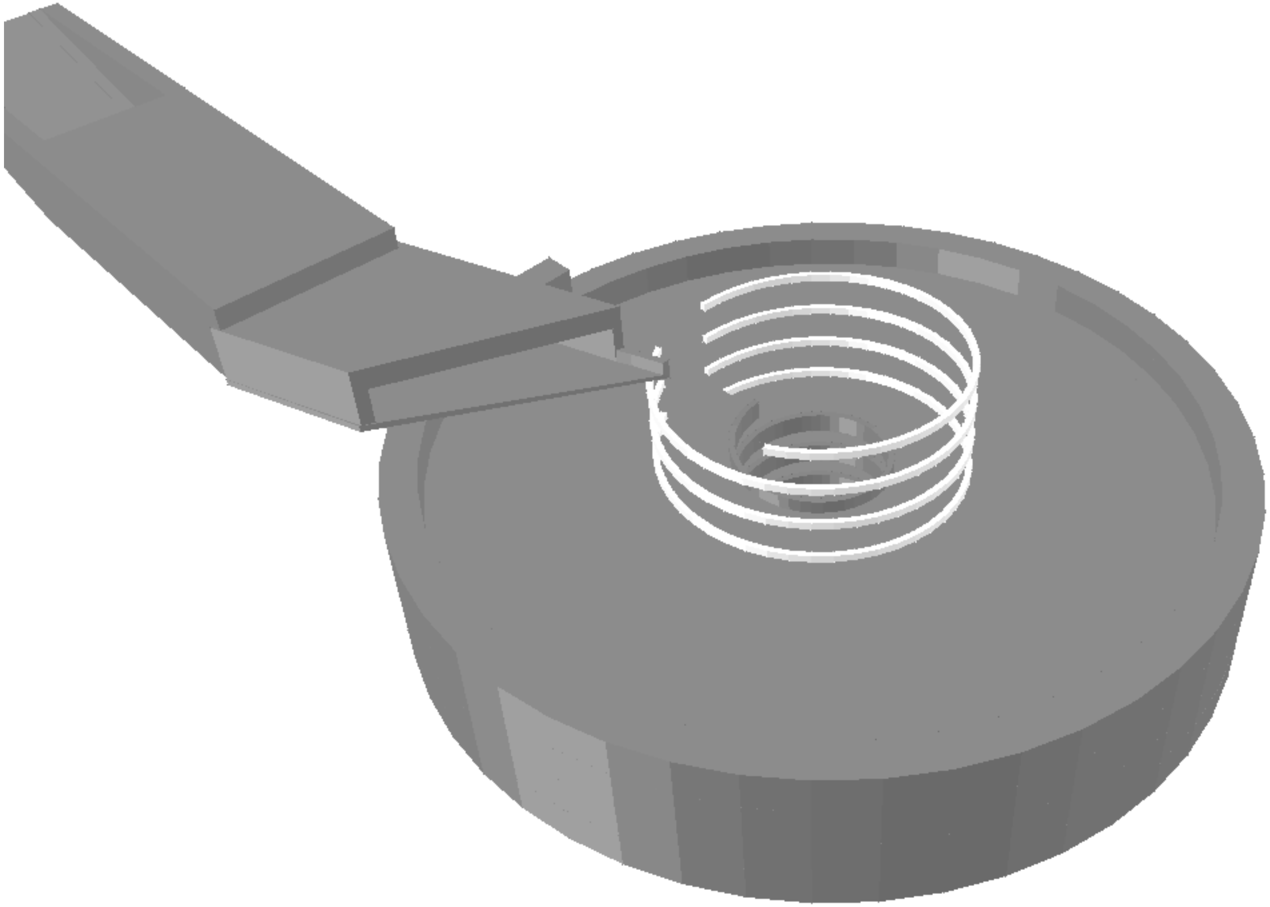


Car Park - Geometry

Seminar

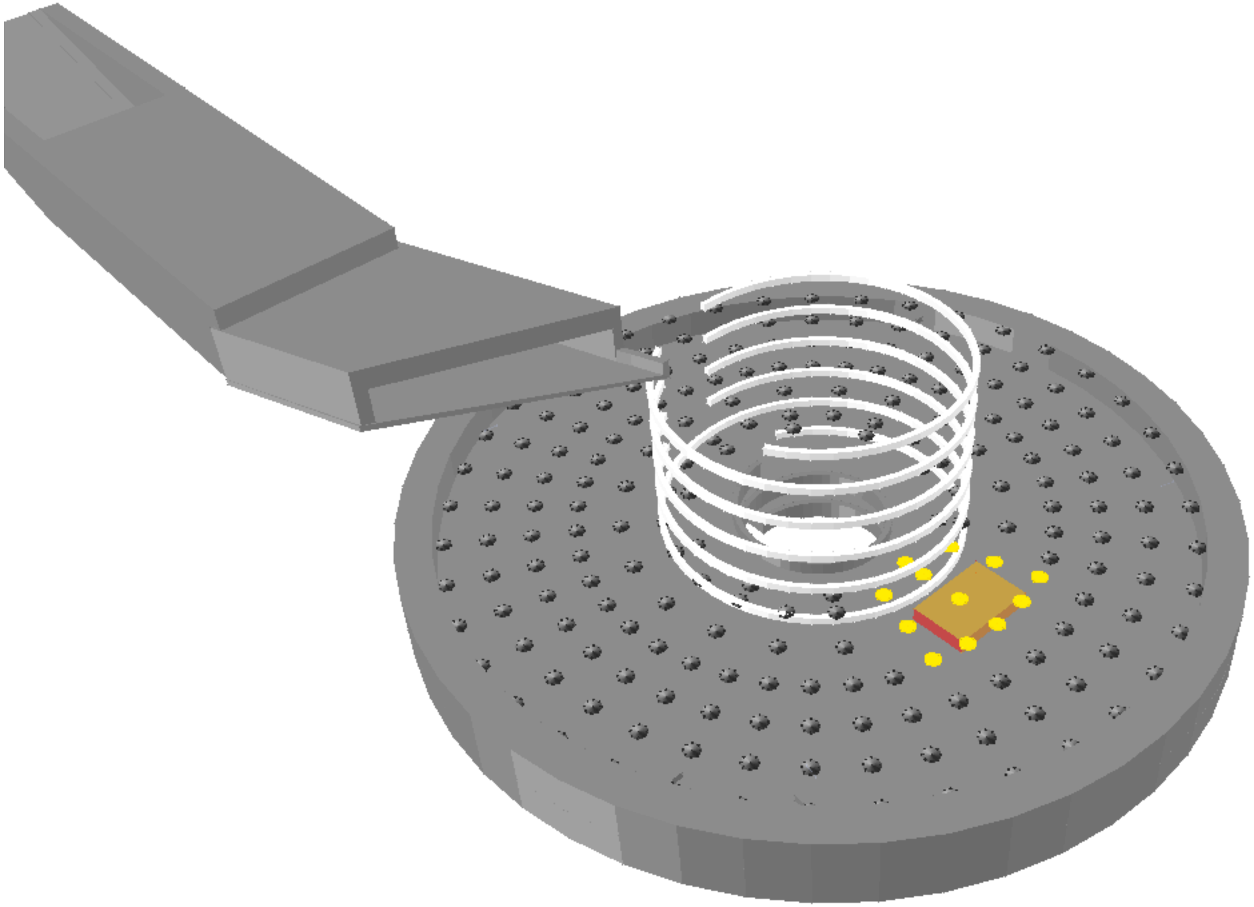
CHAM

- More floors removed

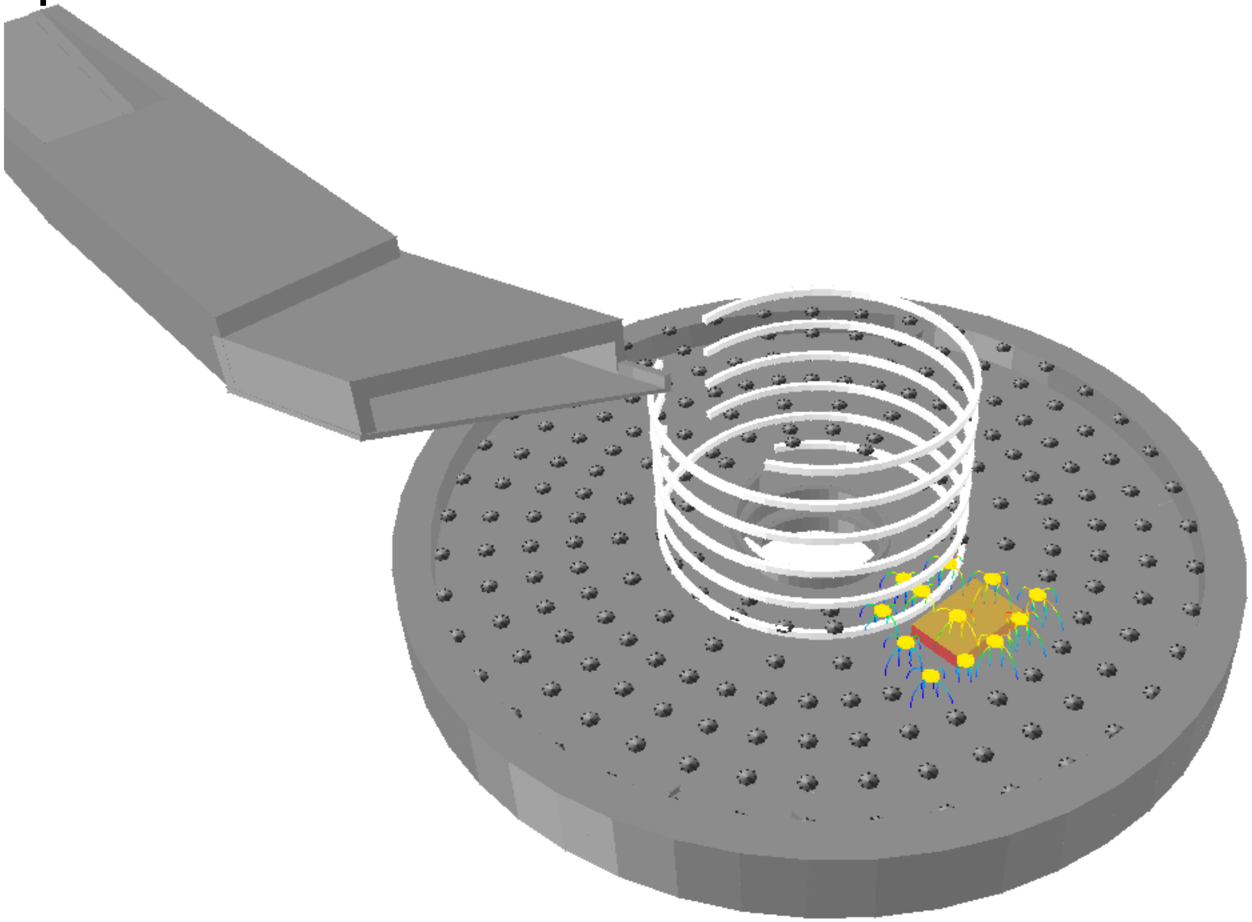


Car Park - Geometry

- Fire and sprinklers on lowest floor

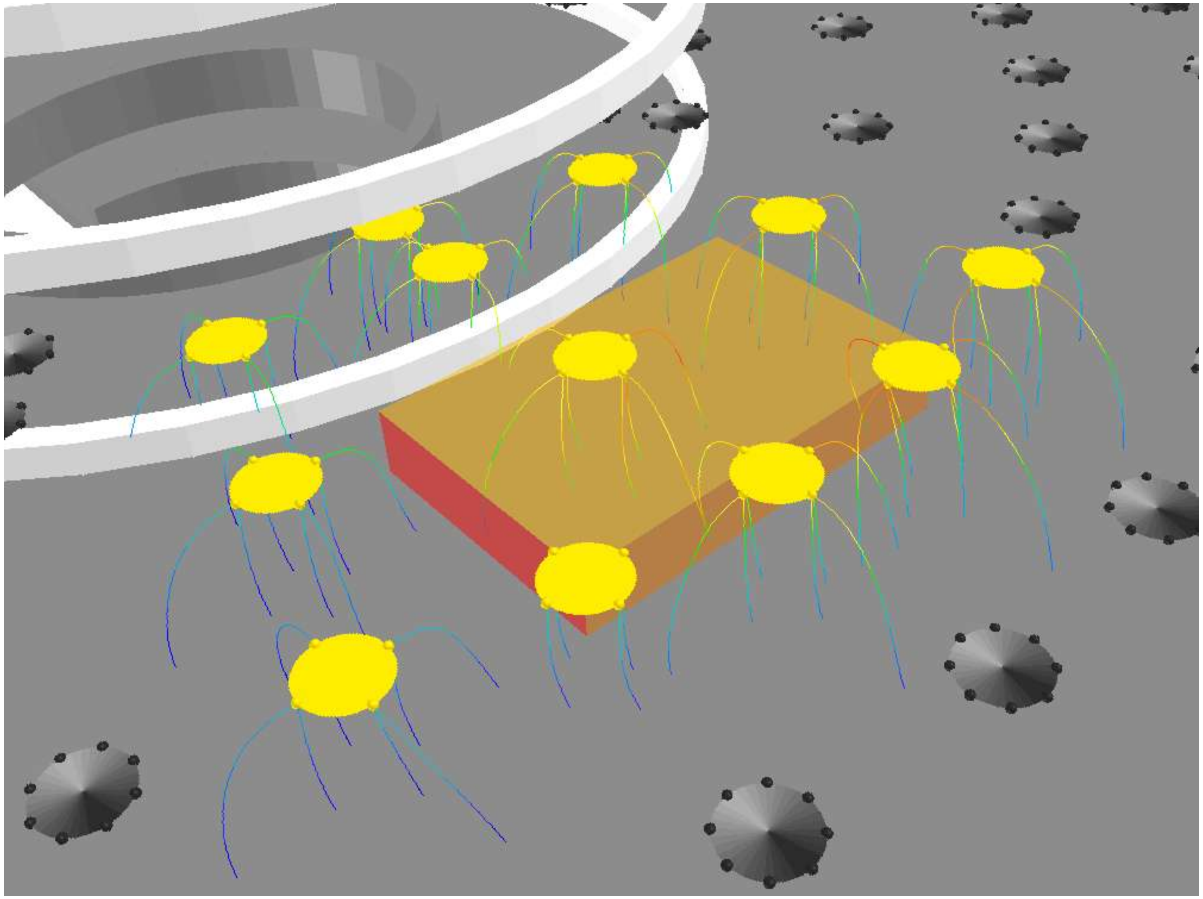


- Sprinklers activated



Car Park - Sprinklers

- Close-up of sprinklers



Car Park - Visibility

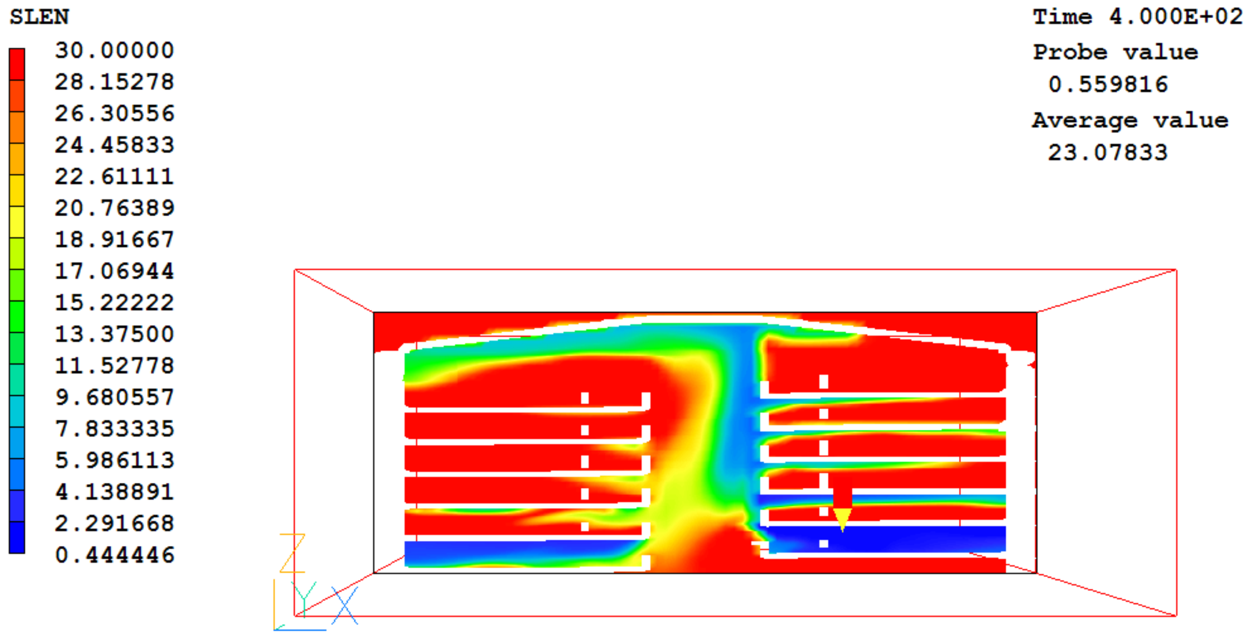
Seminar

CHAM

Sight length after 400s. This is the distance you can see. 30m is assumed to be infinitely far.

The visibility near the fire is poor, <0.5m, but is still acceptable on the upper floors.

The central well is also filling with smoke.



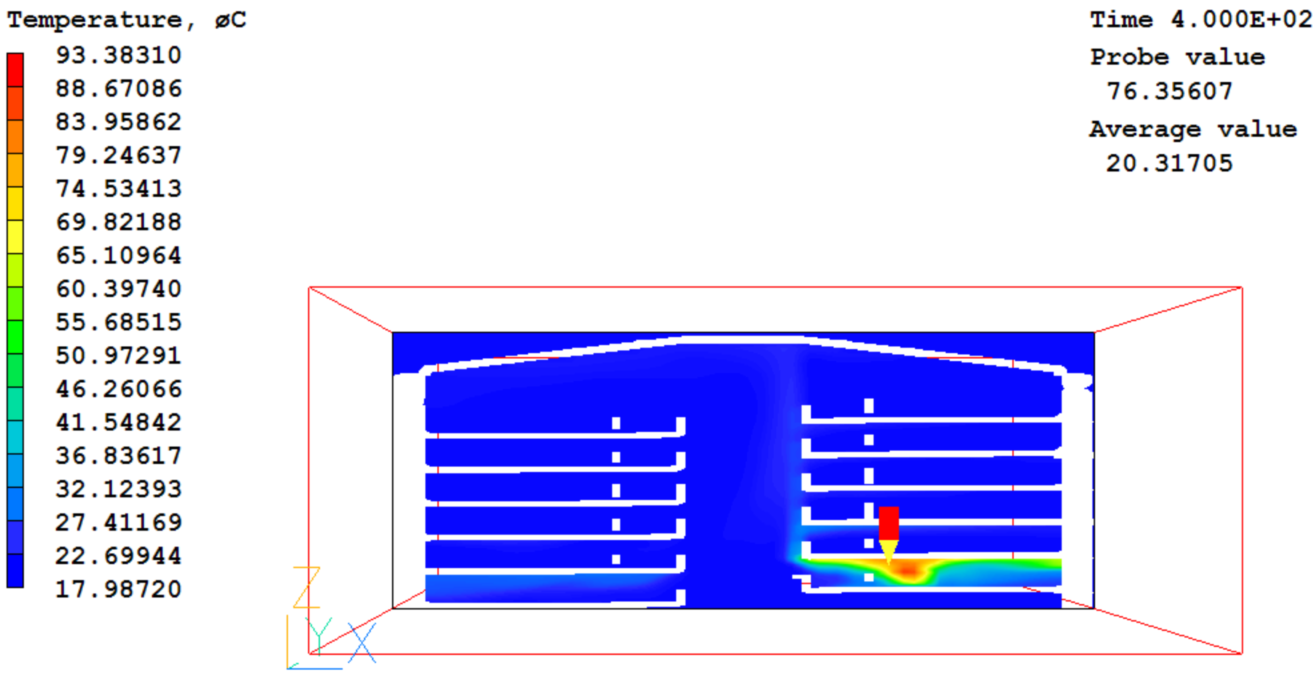
Car Park - Temperature

Seminar

CHAM

This slide shows the temperature contours in a plane near the fire after 400s.

The high temperatures are mainly on the floor with the fire.



Car Park - Visibility

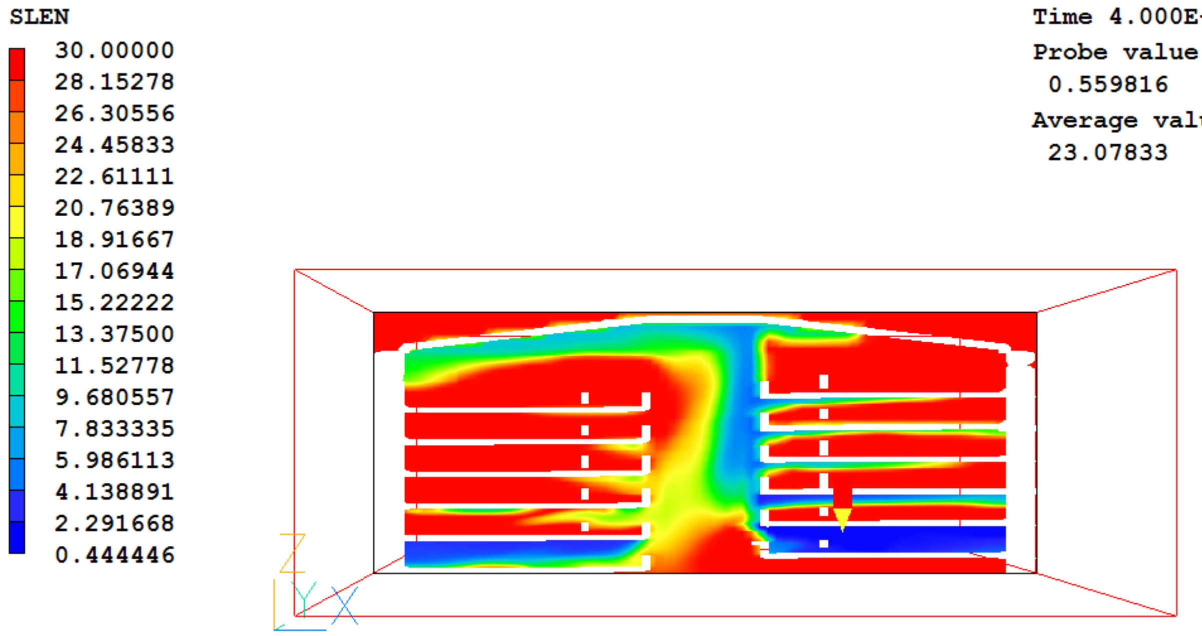
Seminar

CHAM

Sight length after 400s. This is the distance you can see. 30m is assumed to be infinitely far.

The visibility near the fire is poor, <0.5m, but is still acceptable on the upper floors.

The central well is also filling with smoke.



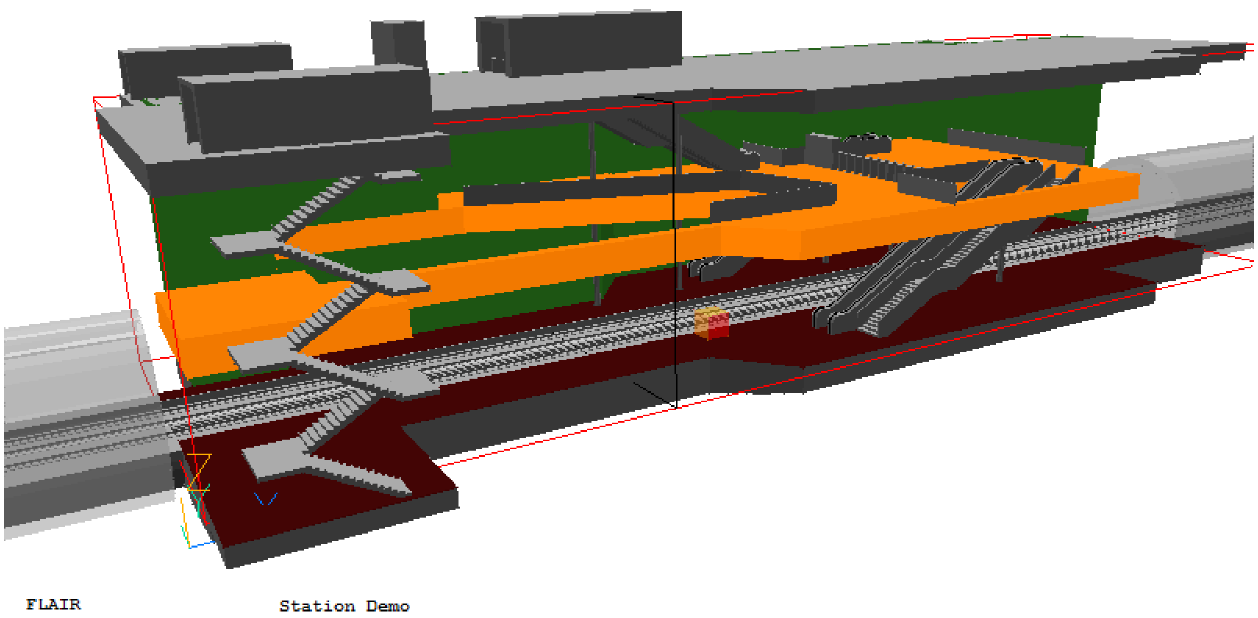
Underground Rail Station Fire

Seminar

CHAM

In the following scenario, the intention is to simulate a fire on an incoming (moving) train as it exits a railway tunnel to a station platform. The conditions are as follows:

The train tunnel is at the bottom of the main domain containing the station and platform areas, with different stairs leading to main lobby upstairs and outside, as well as emergency stairs.



Underground Rail Station Fire

Seminar

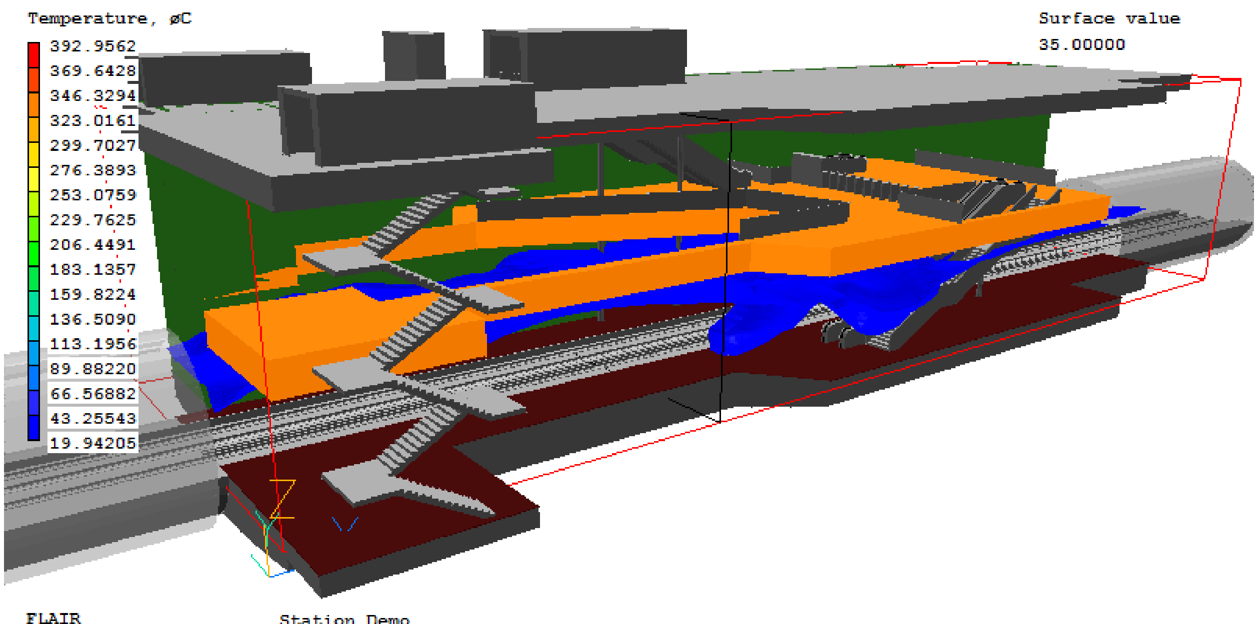
CHAM

The requirement was to simulate a fire in a train that is moving from the tunnel to the station where it stops.

- Fire power goes lineally from 2 MW to 30 MW in 15 minutes.
- The fire is at a ¼ of the train edge
- The tunnel of the train arriving to the station is an inlet of 80 m3/s.

The opposite tunnel is an outlet with the same flow.

- The platform where the train stops has 3 or 4 outlets in each side, totalling 180 m3/s of flow



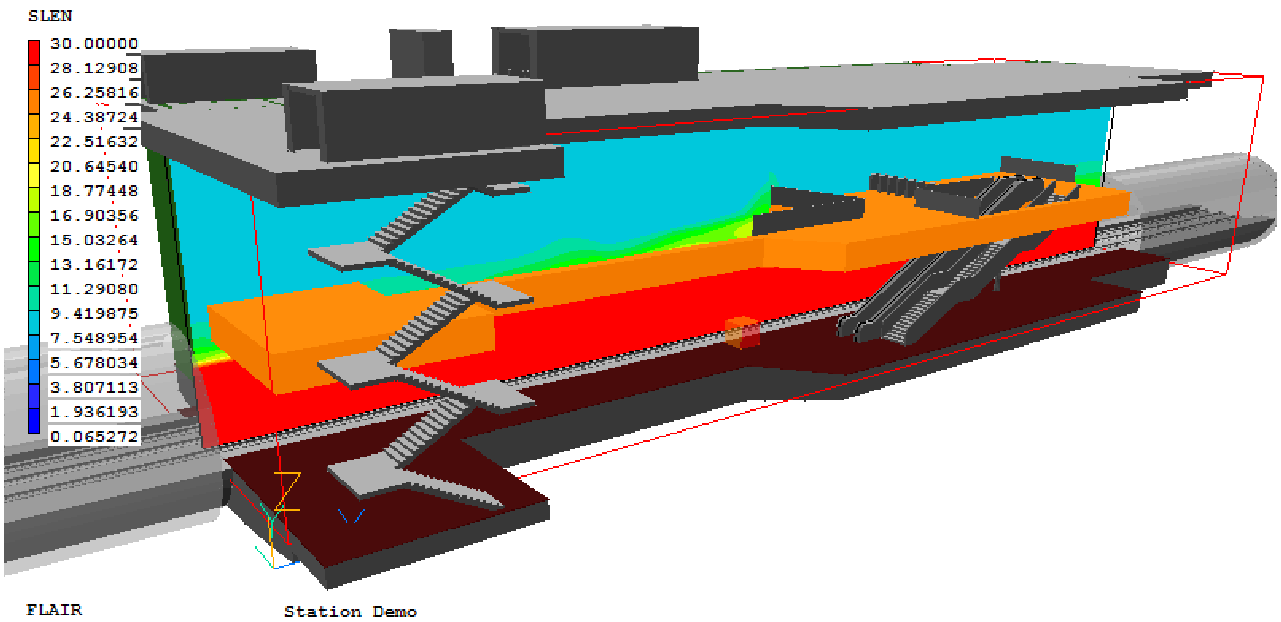
Underground Rail Station Fire

Seminar

CHAM

One stairwell is sealed off and therefore excluded. The outlets are at the top of the domain. A fire (box) has been placed on the platform edge fairly centrally.

The boundary conditions and volume flow rates specified by the client in and out from the tunnel were used, plus an arbitrary default temperature of 20˚C.

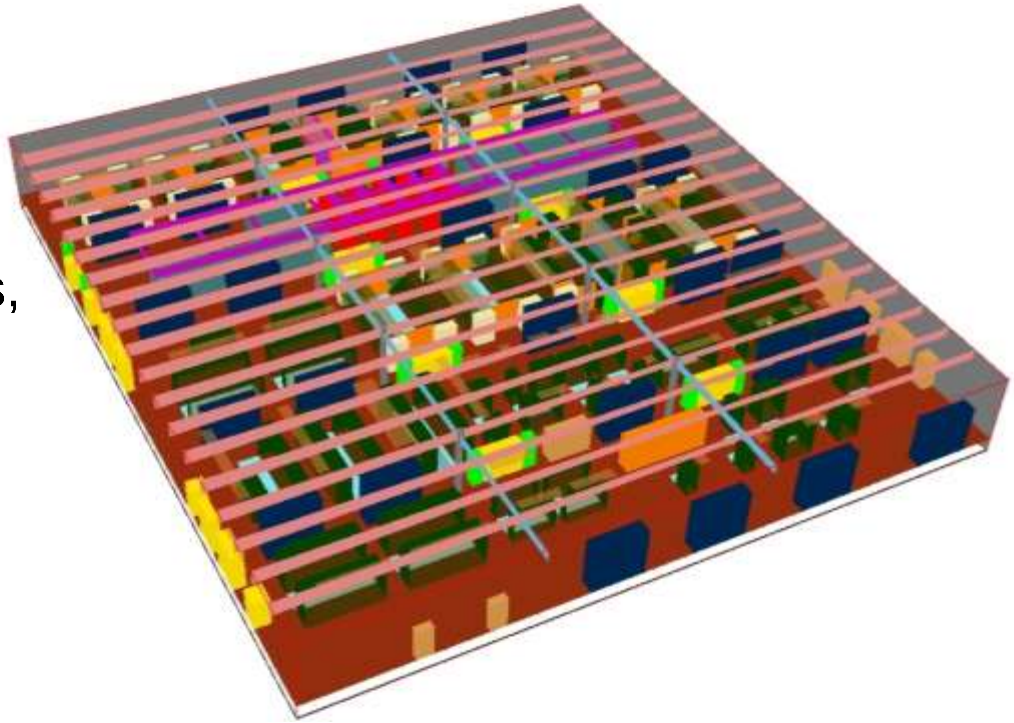


Data Centre Simulation

CHAM

CFD analysis performs a vital role within data centre design, management and operational processes.

CFD helps maximise the performance of cooling and ventilation systems, model the impact of additional loading and equipment distribution, and investigate emergency shut-down scenarios.



Data Centre Simulation

Seminar

CHAM

A streamlined method has been developed at CHAM that constructs a list of data centre contents together with their key parameters (e.g. layout of all cabinets, dimensions, air flow rates, heat output, orientation and other parameters for each one) within a single spreadsheet.

Data Centre Simulation

CHAM

The spreadsheet is read by PHOENICS, enabling common data centre objects (i.e. CRACs, cabinets, floor / ceiling grilles) to be constructed automatically.

This method allows rapid changes to be affected, such as scaling IT loads by changing a single value in the spreadsheet.

Height off Floor (m)		Scale=	0.0254
Xoff (m)	0.508		
Yoff (m)	0		
No of cabinets	43		

Cab No	Width m	Depth m	Height m	Xpos m	Ypos m	Zpos m	FlowRate m3/s	Heat Output W	Heat Load Zpos	Heat Load Height	Orientation NSEW
1	9.75	0.97	2.1336	2.77	4.2672	0	1.74719622	20720	0	2.1336	E
2	9.75	0.97	2.1336	4.85	4.27	0	1.74719622	20720	0	2.1336	E
3	0.61	0.97	2.1336	7.04	4.88	0	0.10919976	1295	0	2.1336	E
4	0.61	0.97	2.1336	7.04	6.10	0	0.10919976	1295	0	2.1336	E
5	6.71	0.97	2.1336	7.04	7.32	0	1.2011974	14245	0	2.1336	E
6	9.75	0.97	2.1336	9.12	4.27	0	1.74719622	20720	0	2.1336	E
7	9.14	0.97	2.1336	11.30	4.88	0	1.63799646	19425	0	2.1336	E
8	9.75	0.97	2.1336	13.39	4.27	0	1.74719622	20720	0	2.1336	E
9	0.61	0.97	2.1336	15.57	5.49	0	0.10919976	1295	0	2.1336	E
10	3.05	0.97	2.1336	15.57	7.32	0	0.54599882	6475	0	2.1336	E
11	1.22	0.97	2.1336	15.57	10.97	0	0.21839953	2590	0	2.1336	E
12	1.22	0.97	2.1336	15.57	12.80	0	0.21839953	2590	0	2.1336	E
13	9.75	0.97	2.1336	17.65	4.27	0	1.74719622	20720	0	2.1336	E
14	9.75	0.97	2.1336	19.84	4.27	0	1.74719622	20720	0	2.1336	E
15	9.75	0.97	2.1336	26.19	4.27	0	1.74719622	20720	0	2.1336	E
16	9.75	0.97	2.1336	28.37	4.27	0	1.74719622	20720	0	2.1336	E
17	9.75	0.97	2.1336	30.45	4.27	0	1.74719622	20720	0	2.1336	E
18	9.75	0.97	2.1336	32.64	4.27	0	1.74719622	20720	0	2.1336	E
19	15.85	0.97	2.1336	36.91	2.44	0	2.83919386	33670	0	2.1336	E
20	3.05	0.97	2.1336	35.08	17.68	0	0.54599882	6475	0	2.1336	E
21	3.05	0.97	2.1336	32.89	17.68	0	0.54599882	6475	0	2.1336	E
22	3.05	0.97	2.1336	30.81	17.68	0	0.54599882	6475	0	2.1336	E
23	10.36	0.97	2.1336	9.47	32.92	0	1.85639599	22015	0	2.1336	E
24	11.58	0.97	2.1336	11.56	32.92	0	2.07479551	24605	0	2.1336	E
25	11.58	0.97	2.1336	14.00	32.92	0	2.07479551	24605	0	2.1336	E
26	9.14	0.97	2.1336	16.18	34.14	0	1.63799646	19425	0	2.1336	E
27	2.44	0.97	2.1336	24.97	40.23	0	0.43679906	5180	0	2.1336	E
28	4.88	0.97	2.1336	26.85	37.80	0	0.87359811	10360	0	2.1336	E
29	4.88	0.97	2.1336	28.63	37.80	0	0.87359811	10360	0	2.1336	E
30	0.61	0.97	2.1336	30.20	36.71	0	0.10919976	1295	0	2.1336	E
31	0.61	0.97	2.1336	30.20	39.93	0	0.10919976	1295	0	2.1336	E
32	0.61	0.97	2.1336	30.20	41.15	0	0.10919976	1295	0	2.1336	E
33	0.61	0.97	2.1336	30.20	42.37	0	0.10919976	1295	0	2.1336	E
34	1.22	0.97	2.1336	32.64	38.40	0	0.21839953	2590	0	2.1336	E
35	0.61	0.97	2.1336	32.64	42.06	0	0.10919976	1295	0	2.1336	E
36	15.85	0.97	2.1336	36.91	28.65	0	2.83919386	33670	0	2.1336	E
37	2.44	0.97	2.1336	35.33	26.21	0	0.43679906	5180	0	2.1336	E
38	2.44	0.97	2.1336	33.50	26.21	0	0.43679906	5180	0	2.1336	E
39	2.44	0.97	2.1336	31.67	26.21	0	0.43679906	5180	0	2.1336	E
40	1.22	0.97	2.1336	32.89	32.31	0	0.21839953	2590	0	2.1336	E
41	1.83	0.97	2.1336	29.85	32.31	0	0.32759929	3885	0	2.1336	E
42	1.83	0.97	2.1336	26.80	32.31	0	0.32759929	3885	0	2.1336	E
43	2.44	0.97	2.1336	35.93	28.96	0	0.43679906	5180	0	2.1336	N
							42.1511069	499870			

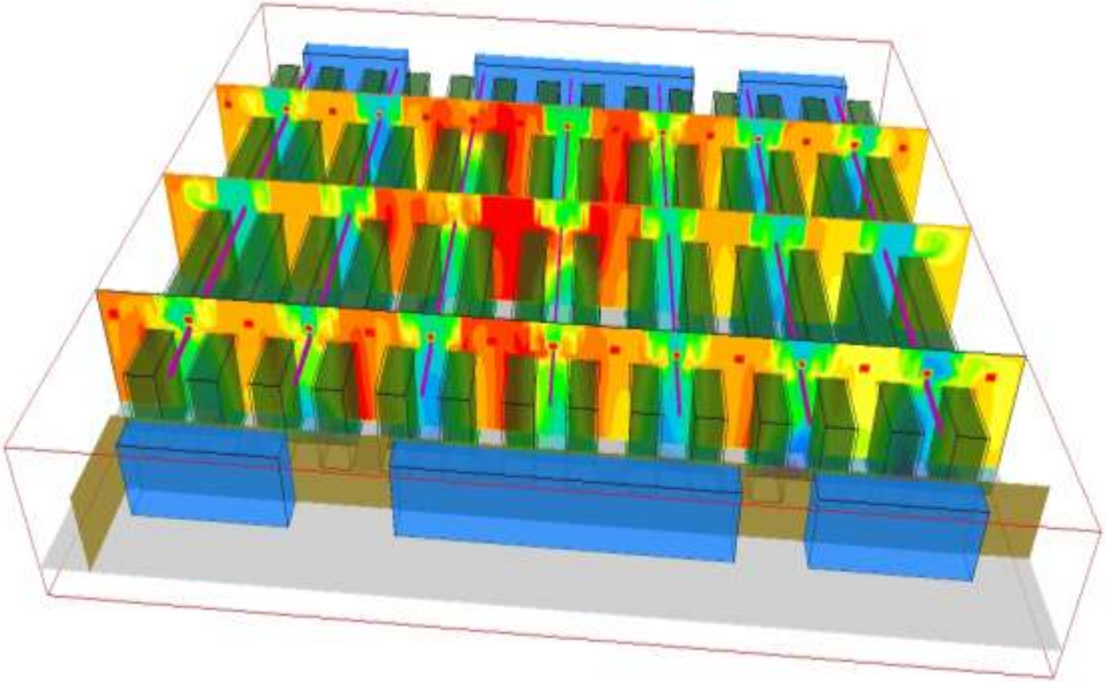
Data Centre Simulation

Seminar

CHAM

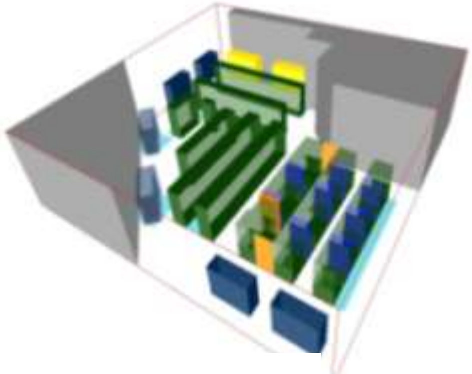
Numerical results are displayed in tabular form with XY plots. In addition, temperature, velocity, humidity and pressure values are displayed in an interactive 3D graphical environment, together with residence-time data streamlines, iso-surfaces and concentration levels.

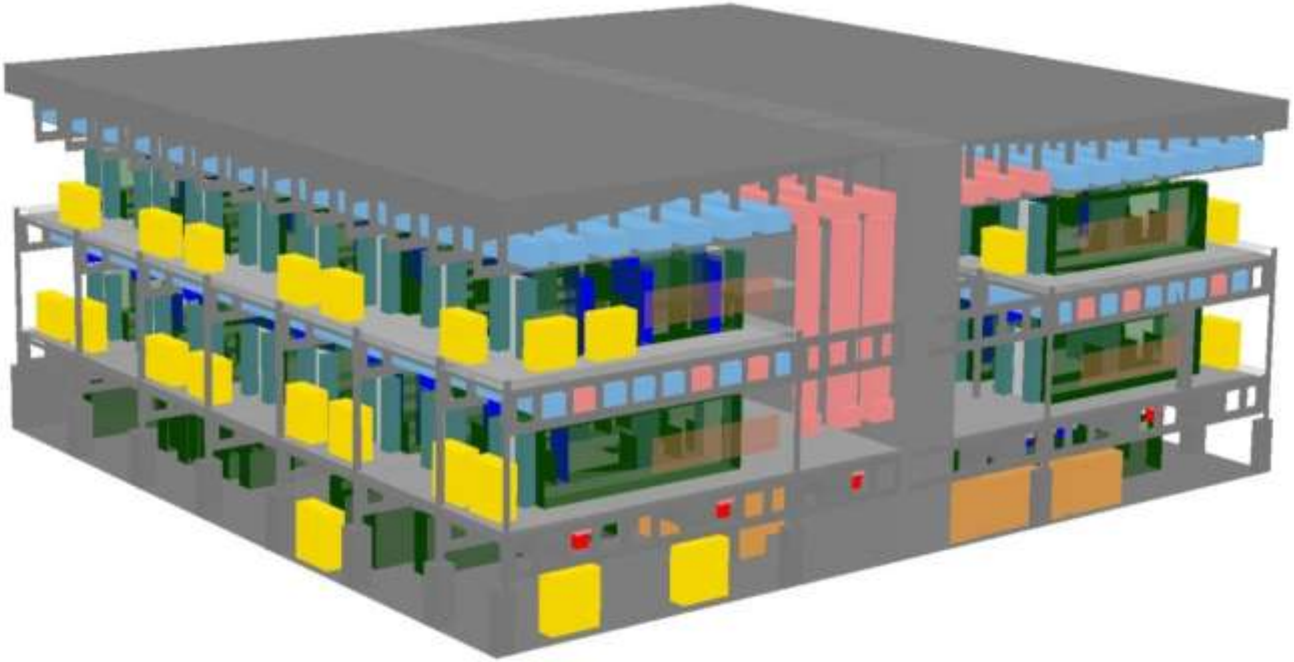
Results can be displayed using either SI or Imperial units.



Data Centre Simulation

PHOENICS/FLAIR handles with ease complex room and equipment layouts, non-standard units, and both multi-room and multi-storey environments.

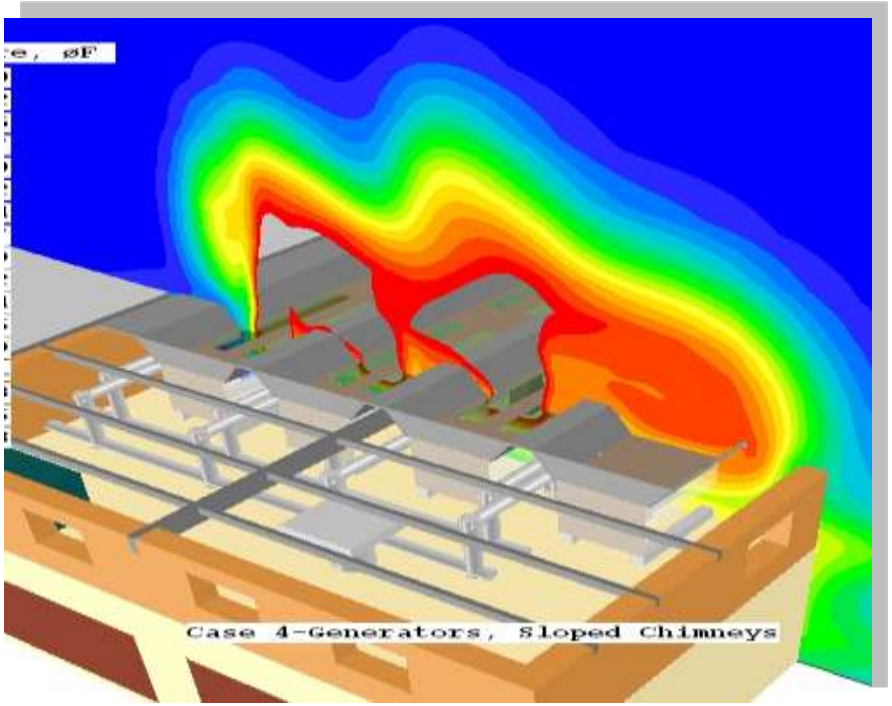




Data Centre Simulation

CHAM

- External influences, such as solar gain, are readily introduced.
- The versatility of PHOENICS/FLAIR is such that it is also appropriate for modelling related equipment, such as the performance of externally-located chilling units subject to the influence of varying environmental conditions, heat extracts from generators and exhaust outlets.

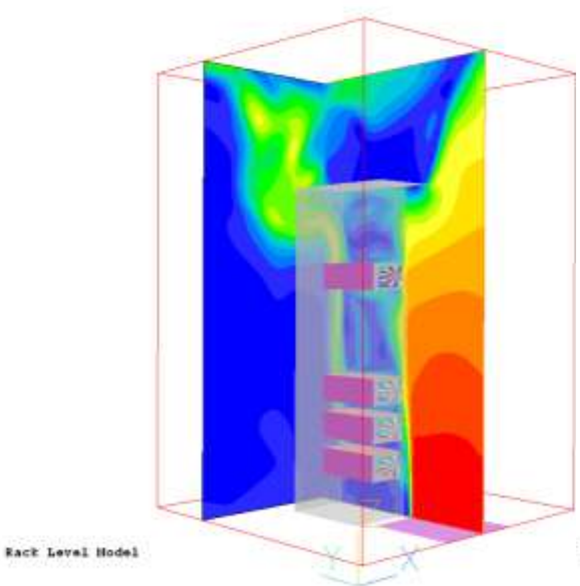


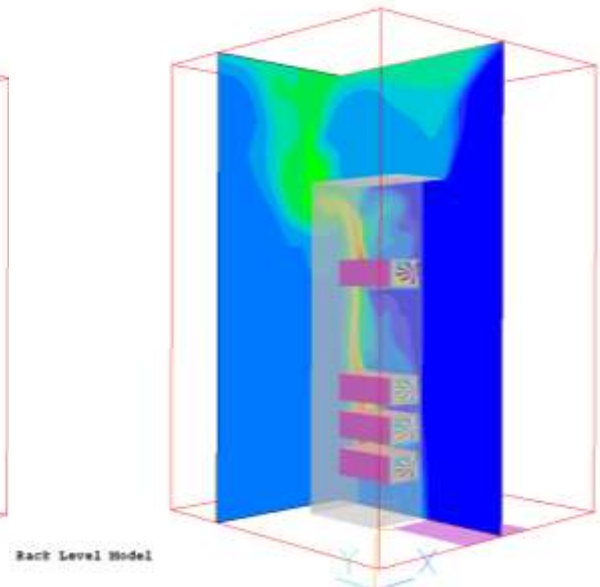
Data Centre Simulation

Seminar

CHAM

Rack-level models for studying and optimising the performance of individual cabinets and their influence upon one another.



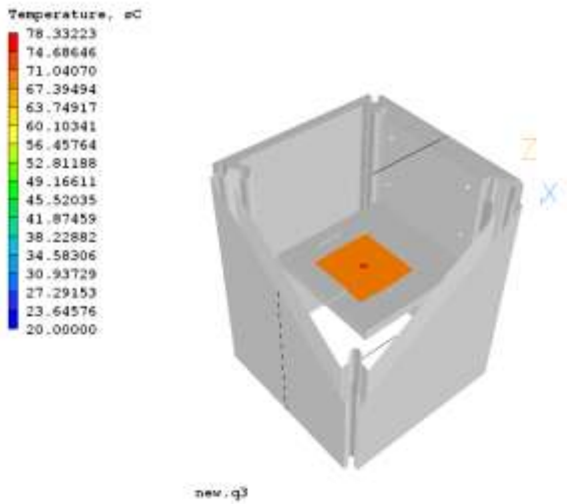


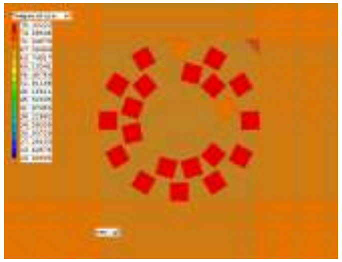
Data Centre Simulation

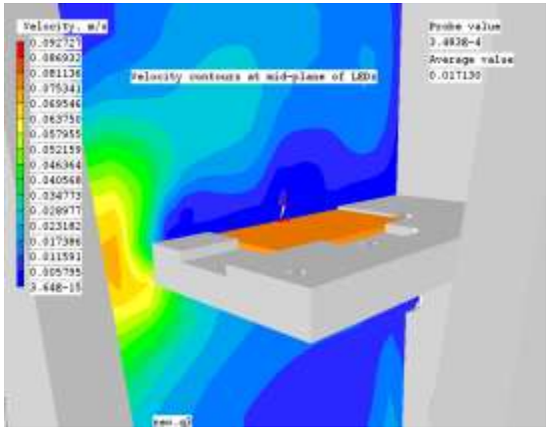
Seminar

CHAM

Ventilation and cooling systems for racks, blades and circuit board LED heat releases can be studied and exported to the larger scale model.







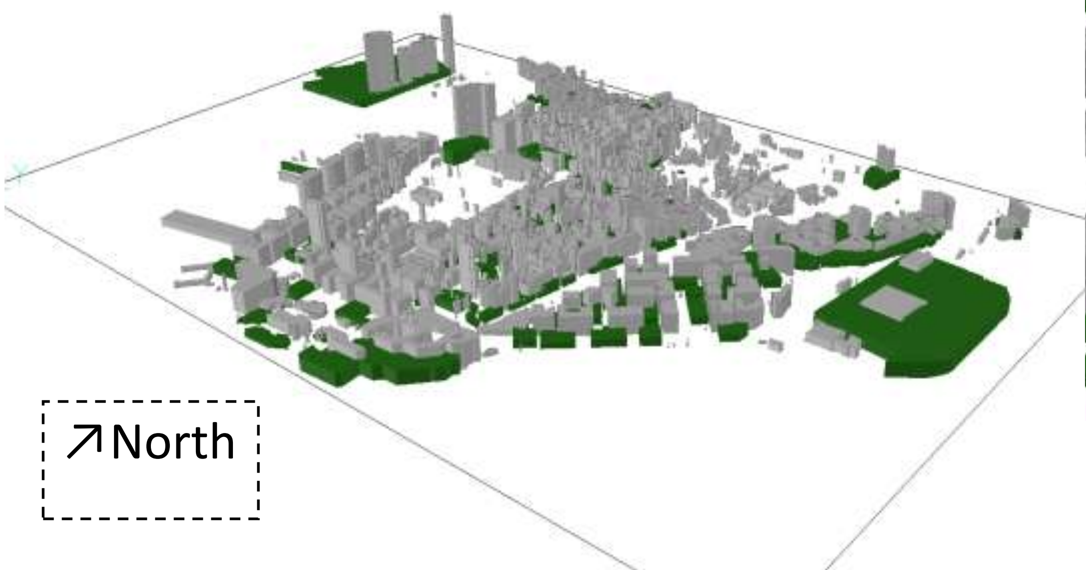
From macro-scale to micro-scale data centre problems, PHOENICS/FLAIR offers a solution.

Urban Wind Flows

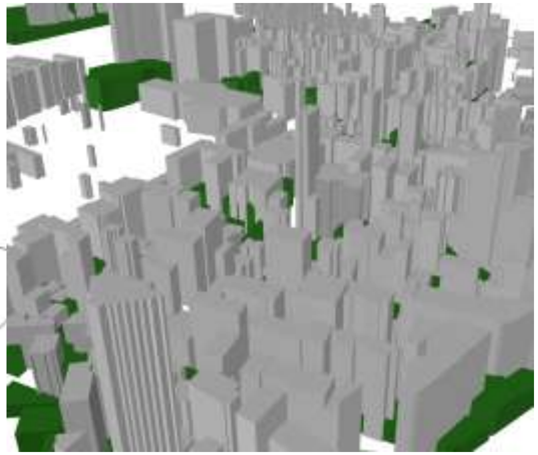
Seminar

CHAM

PHOENICS/FLAIR applied to the simulation of air flows through a cityscape of Hong Kong as part of an Air Ventilation Assessment (AVA) study



Geometry – Cityscape Model



Zoom view

Urban Wind Flows

The building geometry was imported as a DXF file in two interlocking sections (shown as green / grey).

The wind has been specified from the North-East at 10 m/s at a height of 10 metres.

The Domain size was 5H (inflow) x 15H (outflow) x 5H (height) where H=Height of the tallest building at around 250m.

Normalised wind data was supplied by the client. When the data cannot be fitted by logarithmic or power-law expressions, the data can be introduced to PHOENICS/FLAIR via “INFORM” – a facility for introducing any user-defined formulae.

Urban Wind Flows

Physical Domain size: 2587m * 1711m * 1000m (roughly 10H * 7H * 4H).

Grid size: 744 * 704 * 45 (23.57 million cells). The grid was divided into 3 regions in X and Y. The central region of 750m was assigned 500 cells, giving a cell-size of 1.5m. Outside the central zone, the grid was allowed to expand towards the domain boundaries.

Wind Velocity: The wind speed at 10m was taken to be 10m/s, from the Northeast. A logarithmic profile was used with a reference roughness height of 0.25m (equivalent to ‘scattered obstacles’).

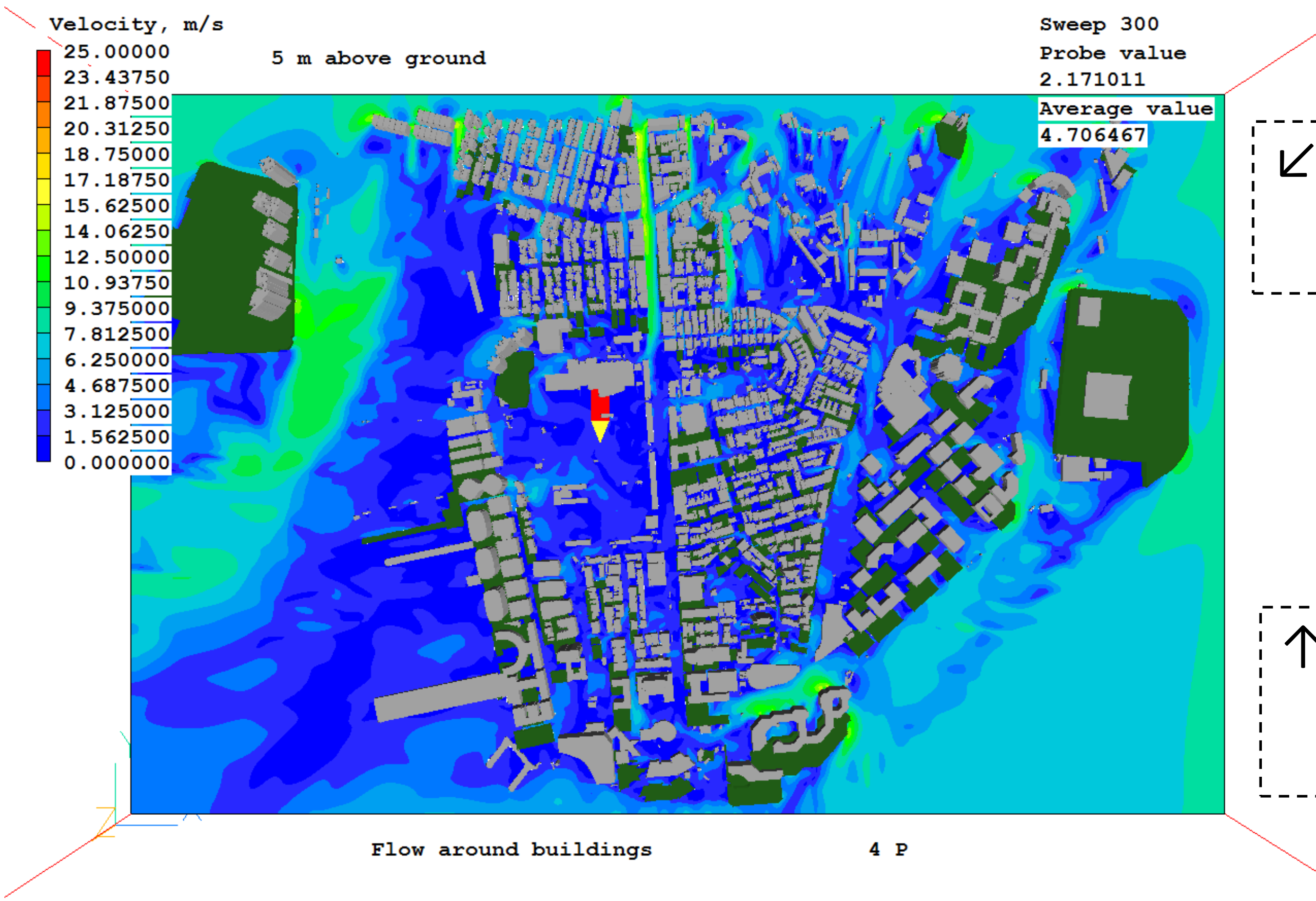
The geometry had been exported from AutoCAD as a DXF file containing the surfaces of the roof tops. This file was used in AC3D to extrude down to the ground plane thus creating the closed volumes required by PHOENICS.

Urban Wind Flows

Seminar

CHAM

Wind velocity contours @ 5m above ground



↙Wind

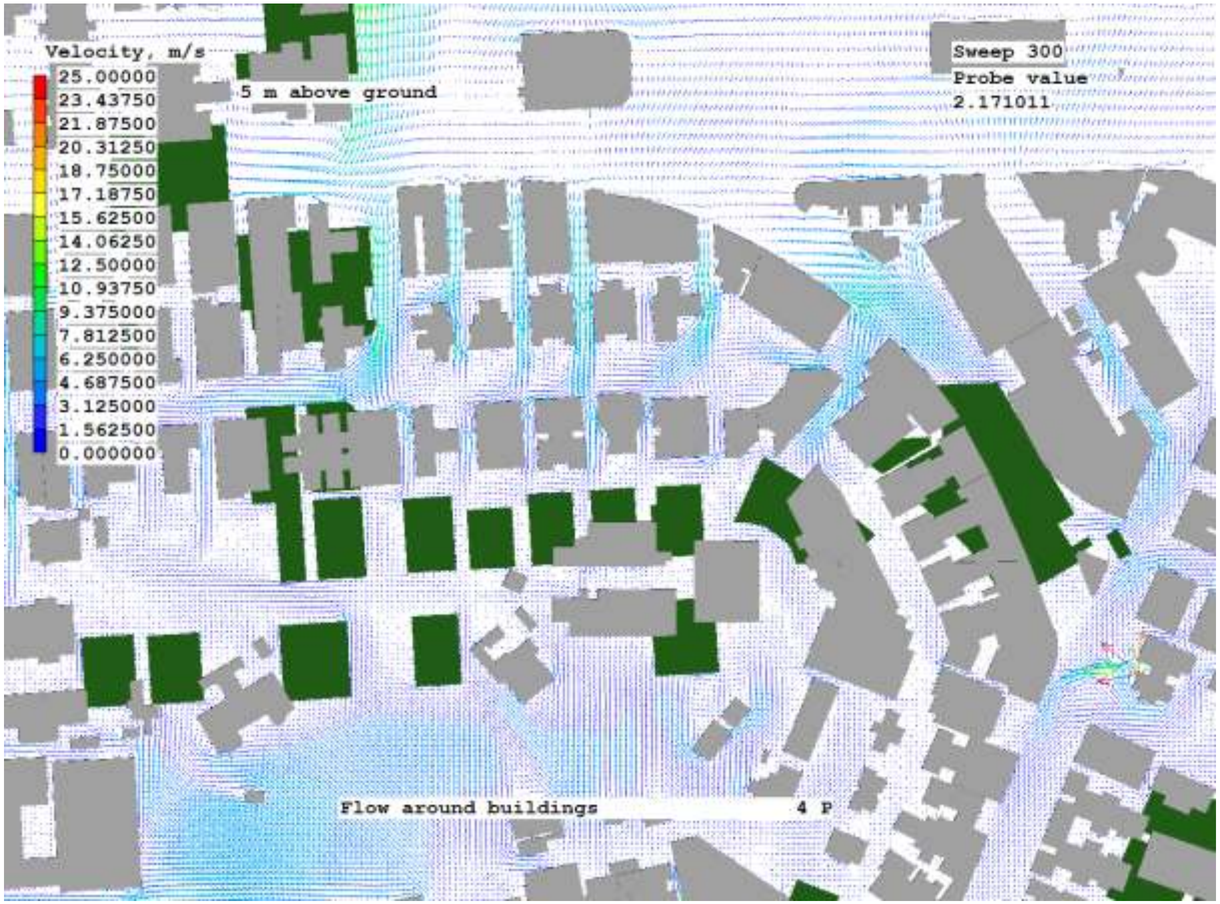
↑North

Urban Wind Flows

Seminar

CHAM

Zoom image - Wind velocity vectors @ 5m above ground



↙Wind

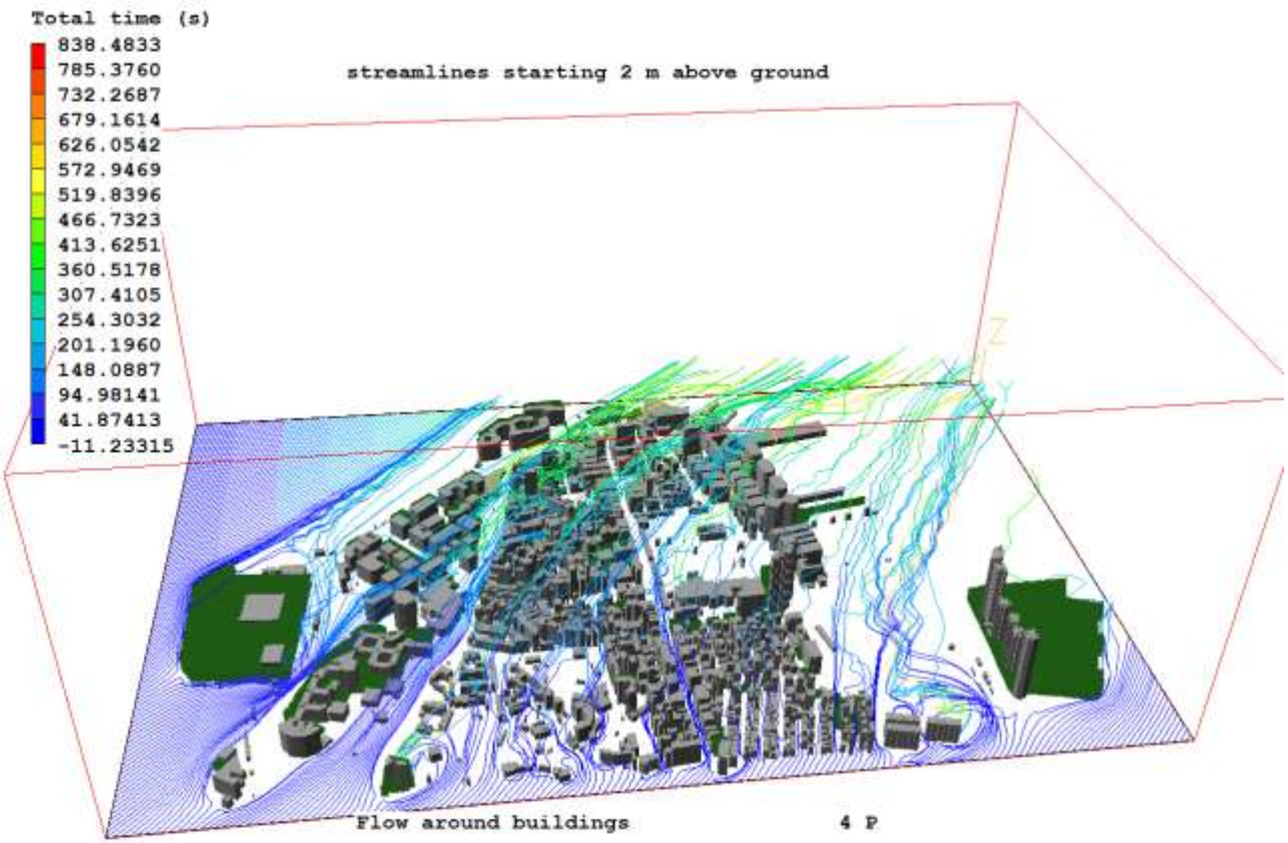
↑North

Urban Wind Flows

Seminar

CHAM

Streamlines starting @ 2m above ground



↗Wind

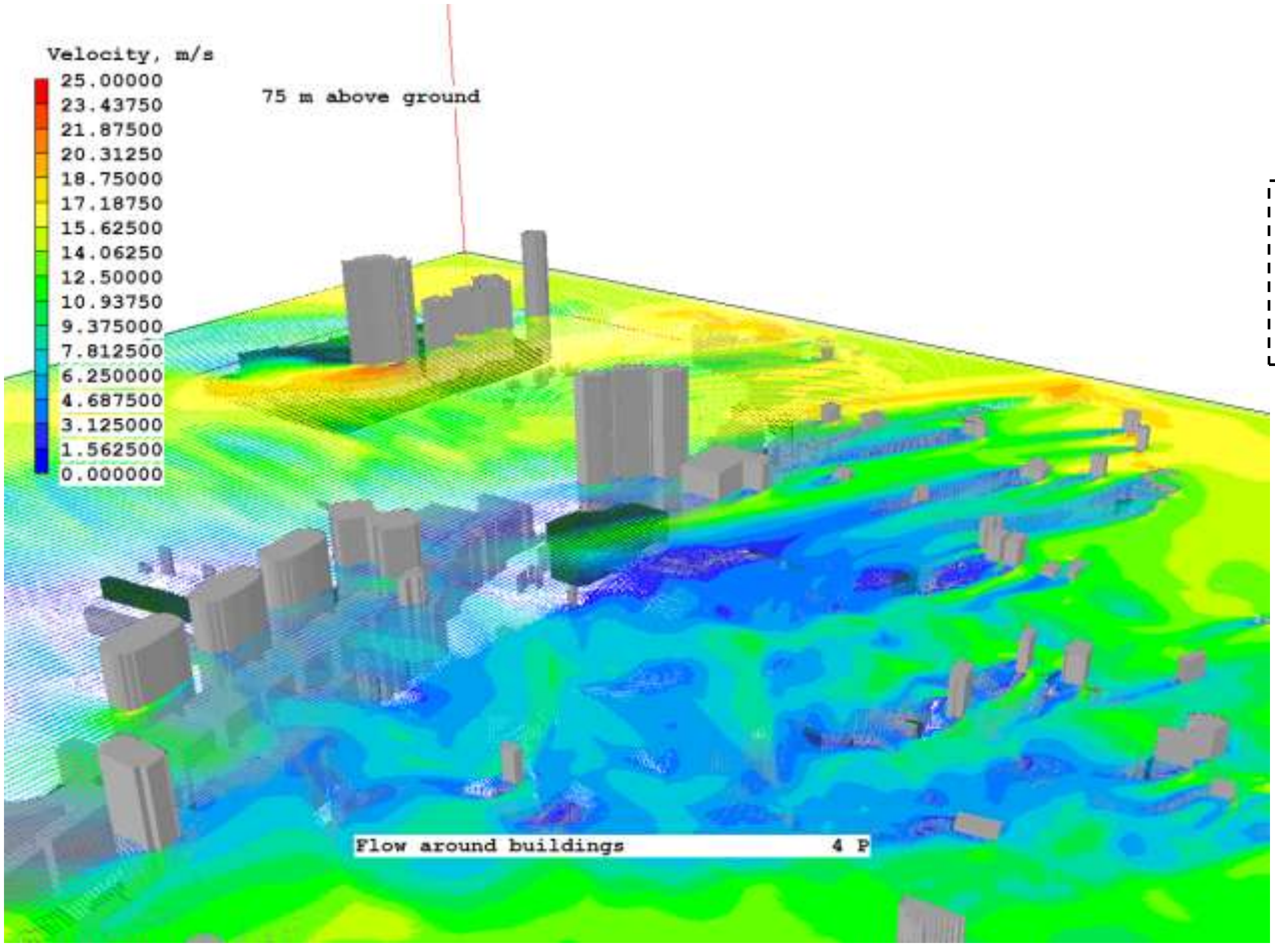
↓North

Urban Wind Flows

Seminar

CHAM

Zoom image - Wind velocity vectors @ 75m above ground



←Wind

↗North

Urban Wind Flows

The results readily show the variation in wind speed through the street canyons, areas of high turbulence, and calm regions. The plots shown were generated using the standard VR-Viewer post-processor which can display vectors, contours, iso-surfaces, high & low spots, and animated view options.

In the same way that alternative CAD products can be used for geometry creation and import into PHOENICS, users also have the option to export the results to third-party post-processors [such as FEMView, Fieldview, GLView, Paraview, TECPLOT and Wavefront] for which interfaces are available within PHOENICS.

Road Tunnel
Nox Reduction

Air pollution from vehicle exhausts is a serious public health issue, particularly in road tunnels where pollutants may be concentrated at harmful levels.

Long tunnels may require significant forced ventilation to keep pollution below maximum safe levels, with tunnel portal geometry and surrounding topography also affecting the dispersion of pollutants.

The tunnel shown in this example is a single bore design with a central partition to separate two lanes of opposing traffic flow.

Despite using ventilation fans to draw air through each lane of the tunnel NO2 concentrations were found to be higher than expected due to re-ingestion of the exhaust flow from one lane back into the tunnel in the opposite lane.

Road Tunnel
Nox Reduction

A PHOENICS model of the tunnel was constructed using CAD of the tunnel geometry and topography surrounding the portals.

The model included ventilation fans within the tunnel and sources to represent vehicle drag and NO2 production.

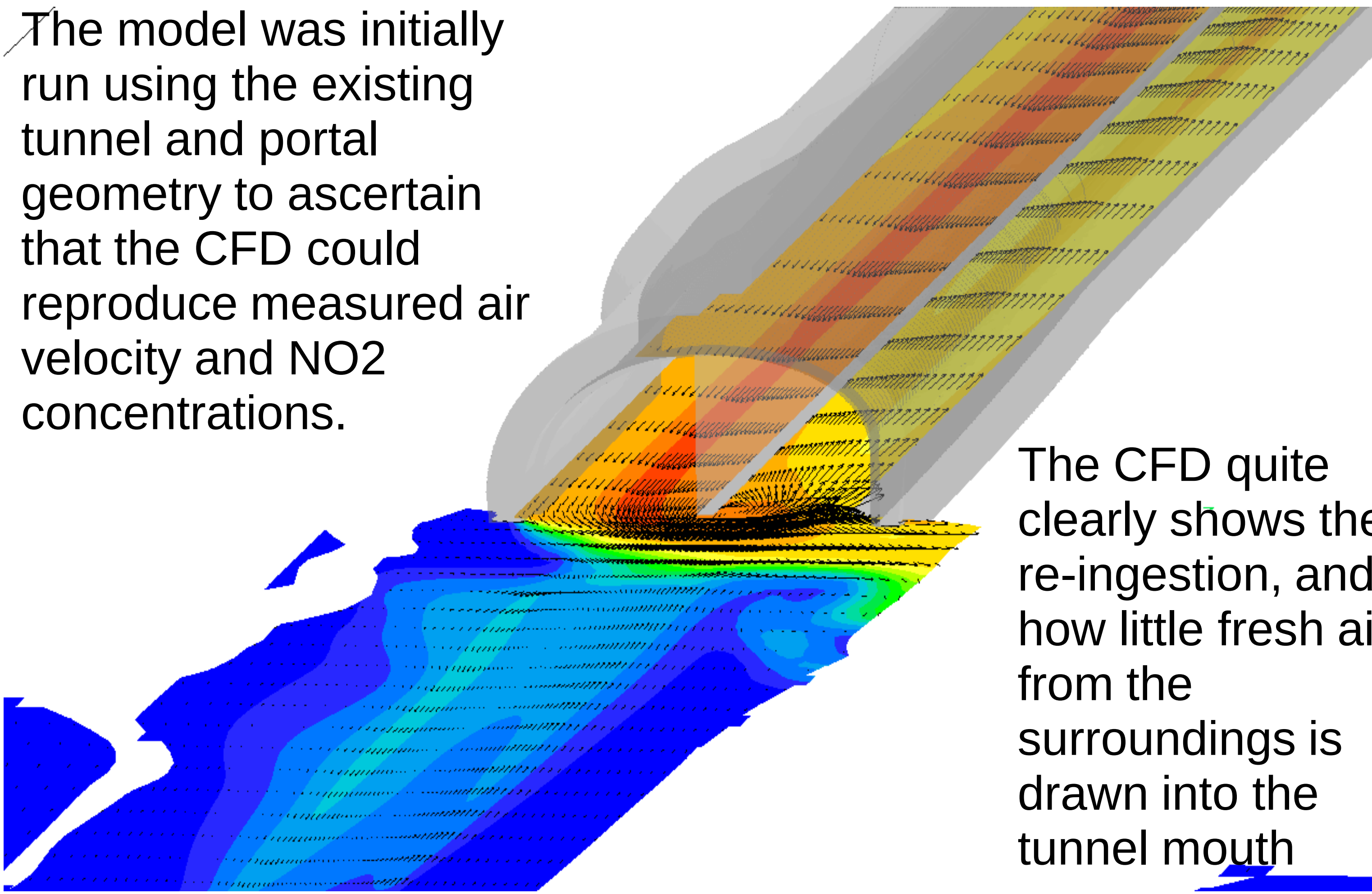
Local wind conditions around the tunnel site were also included in the model.

NO2 concentration
Existing portal design

Seminar

CHAM

The model was initially run using the existing tunnel and portal geometry to ascertain that the CFD could reproduce measured air velocity and NO2 concentrations.

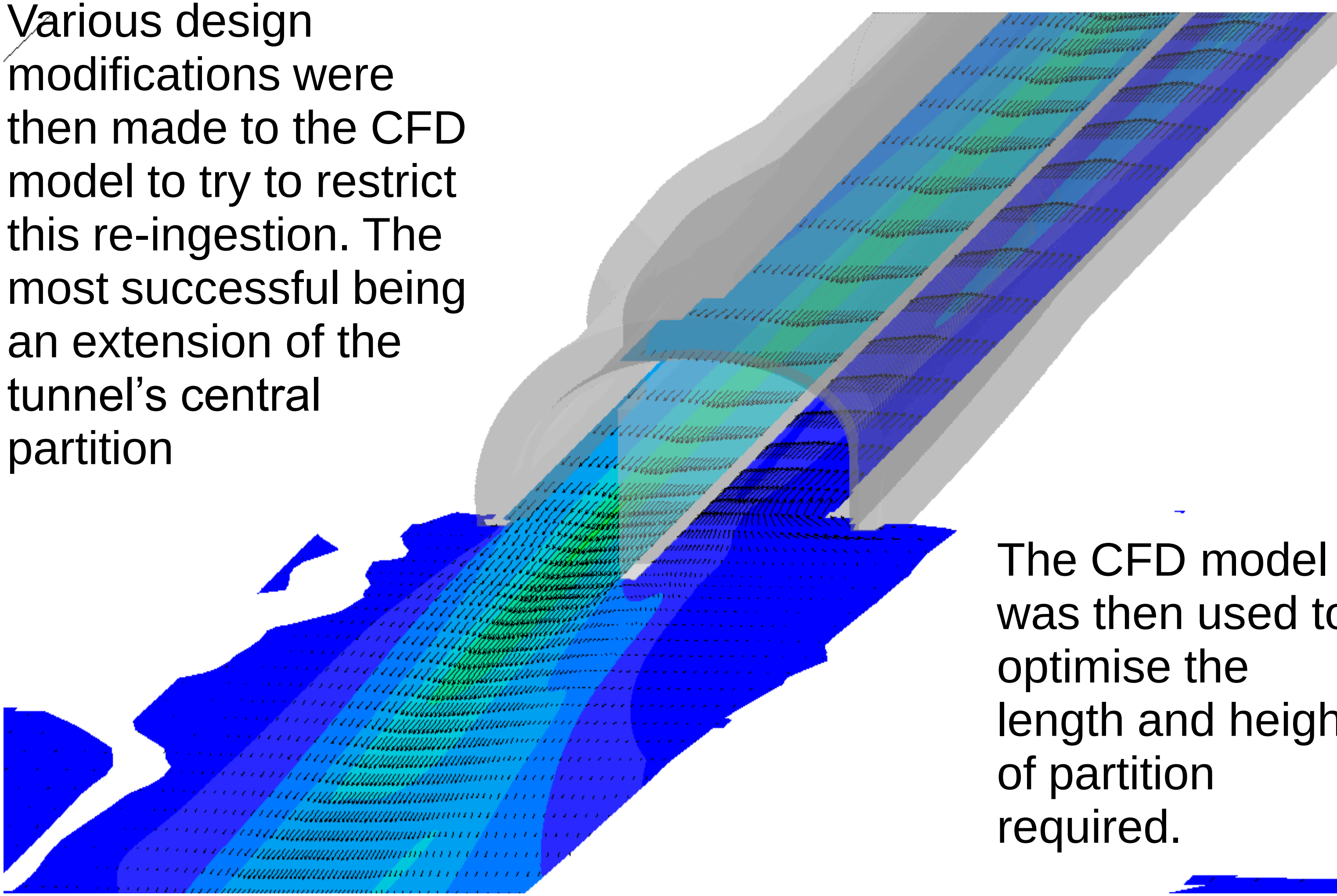


The CFD quite clearly shows the re-ingestion, and how little fresh air from the surroundings is drawn into the tunnel mouth

NO2 concentration
Extended partition

CHAM

Various design modifications were then made to the CFD model to try to restrict this re-ingestion. The most successful being an extension of the tunnel's central partition



The CFD model was then used to optimise the length and height of partition required.

For Information

Seminar

CHAM

For further information about these, and further applications of PHOENICS CFD software, contact CHAM at:

CHAM Ltd
40 High Street
Wimbledon Village
London SW19 5AU
Tel: +44 (0)20 8947 7651
Fax: +44 (0)20 8879 3497
Email: phoenics@cham.co.uk
Web: www.cham.co.uk